Draft Motor Vehicle Emission Simulator (MOVES) 2009

User Guide

Draft Motor Vehicle Emission Simulator (MOVES) 2009

User Guide

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

Table of Contents

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 from gasoline to 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 and/or emission rates. The actual specification of the characteristics of a particular scenario to be modeled is done by creating a Run Specification, or RunSpec.

The MOVES model is different from previous EPA 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 by the user of data specific to a user's unique needs. With its focus on the data, the MOVES model does not require a system rebuild in order to use the most recent data for estimating emissions.

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 Draft MOVES2009

Draft MOVES2009 is a draft version of the MOVES emissions modeling tool. Draft MOVES2009 builds on the functionality of previous MOVES versions: MOVES2004 and the MOVESDemo. Draft MOVES2009 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, Draft MOVES2009 can make projections for energy consumption (total, petroleum-based, and fossil-based). Because the MOVES team lacked the time to update the assumptions, in Draft MOVES2009, life cycle effects (i.e., well-to-

pump emissions) have been set to zero for energy consumption and emissions; and the model no longer includes an interface to the Argonne National Laboratory GREET model.

The draft version of MOVES2009 is meant for user comment and is not intended for official use. Please see the Draft MOVES2009 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 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 and to operate at smaller scales 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. MySQL AB produces 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. Draft MOVES2009 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. Draft MOVES2009 is capable of modeling emissions for the calendar years 1990 and 1999-2050.

MOVES may be operated on a single computer system or on a network of computers. See the Installation Guide and/or the Software Design Reference Manual for more information about specific requirements and computer configurations.

1.2 DRAFT MOVES2009 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 DRAFT MOVES2009 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 DRAFT MOVES2009 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 DRAFT MOVES2009.

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

Software Design: The MOVES Software Design and Reference Manual (SDRM) is available as a stand-alone document and is accessible via the **Help** menu item on the MOVES Main Menu. 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 file in the database itself.

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 DRAFT MOVES2009.

⇒   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 DRAFT MOVES2009 Run Specification" included in Section 1.4 of this guide for instructions.

⇒   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 via the **Help** menu.

⇒   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.

$\Rightarrow$ If you would like to work through an example, a set of sample scenarios is included in the Appendices.

$\Rightarrow$ If you would like to get started using MOVES for regular work, advice on "Best Practices" is available in an Appendix.

The rest of this guide provides information on using the DRAFT MOVES2009 model.

1.3 Installation

Step-by-step instructions on installation are supplied by EPA in the README file in the DRAFT MOVES2009 Software Distribution Suite. (A more detailed discussion is also available in the MOVES Software Design and Reference Manual.) The DRAFT MOVES2009 Software Distribution Suite consists of three required installation programs (MySQL 5.0, Java 1.4.2, and Draft MOVES2009), and several additional programs that can be used to install additional, optional software tools for use with MOVES. Please refer to the README file first if DRAFT MOVES2009 has not been installed on your computer.

1.3.1 System Requirements

DRAFT MOVES2009 has been developed to work only in a Microsoft Windows 2000 and later environment. Using MOVES with Microsoft Vista requires changes to security settings if using the 32-bit architecture; the VISTA 64-bit architecture is not 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) of one GHz or more are highly recommended.

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

The MOVESDefault database distributed with DRAFT MOVES2009 requires approximately 1.3 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 or the MOVES Worker program. Extensive users of MOVES will want to use late-model, high-performance microcomputer systems.

1.3.2 Starting and Testing Your Installation

After all necessary installation packages have been executed, test the installation with the following steps.

1. Check that the MySQL server is operating on all computer systems in the configurations that require it. 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. On most versions of Windows, the MySQL server can be set up to run automatically as a system service. Detailed instructions as to how to do this are contained in the Installation Guide.

2. Double-click the **MOVES Master** program icon if working on a single computer, or if working on the main computer in a computer network.

    Double-click the **MOVES Worker** program icon if using a computer in a multiple-computer configuration that is not the main computer. On multiple-computer configurations these MOVES Worker programs are usually left running indefinitely.

This program does not have to be on the same computer as the MOVES Master Program, but must have access to the shared file directory

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

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. This panel will appear only at start-up.

4. Execute the Example Run Specification (MOVES2009Example.mrs) as described in the following section.

1.4 Executing Example DRAFT MOVES2009 Run Specification

The example DRAFT MOVES2009 Run Specification 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

Assuming that MySQL and DRAFT MOVES2009 have been installed, that the MySQL server and the **MOVES Master** program have been started, and that **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 **MOVES2009Example.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.

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 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 or by viewing the database directly using MYSQL commands, either through the command prompt window or through MySQL Query Browser.

1.4.2 MOVES Output Structure

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

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

2. **MovesActivityOutput:** This table contains the VMT results of the run disaggregated by several operating parameters such as Year, Month, Day, County, Fuel, Model-Year, Road-type, and SCC.

3. **MovesRun:** This table contains information about the date and time of the run 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 diagnostics that might occur if the MOVES run is unsuccessful.

5. **MovesWorkersUsed:** This table contains information as to which MOVES Worker Program copies processed portions of the run, and of interest to most users only if the run is executed on a multi-computer configuration.

6. **Activity Type:** In DRAFT MOVES2009, this table contains only one activity type, distance traveled. Other activity types, such as starts, source hours, or extended idle hours are planned for inclusion in future MOVES versions.

7. **MovesEventLog:** This table is used by MOVES programmers for storing diagnostic results and contains no useful user information.

8. **MovesLookupActivityOutput:** This table contains the distance results from choosing the Emission Rate Calculation Type in the Scale panel in units of distance

as chosen by the user in the General Output panel. These are normalized values and not intended to represent the actual distance in the scenario.

9. **MovesLookupOutputTemp:** This is an interim table of emission rates and is used to calculate rates in the MovesLookupOutput table.

10. **MovesLookupOutput:** This table only appears if you choose the emission rate calculation type from the Scale panel. In this table emissions are reported in units of mass divided by the units of distance as chosen by the user in the General Output panel.

11. **MovesTablesUsed:** This table contains a list of the tables used when executing the RunSpec and the source database for each table.

2.0 Using DRAFT MOVES2009

Using DRAFT MOVES2009, the user can model various scenarios in order to determine specific emissions profiles. DRAFT MOVES2009 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" 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 advanced technologies or fuel scenarios.

Several components of the model are intended to facilitate customization. These include the **I/M Coverage Table Editor** described in Appendix A and 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.

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.

Main Menu Bar

Detail Panel

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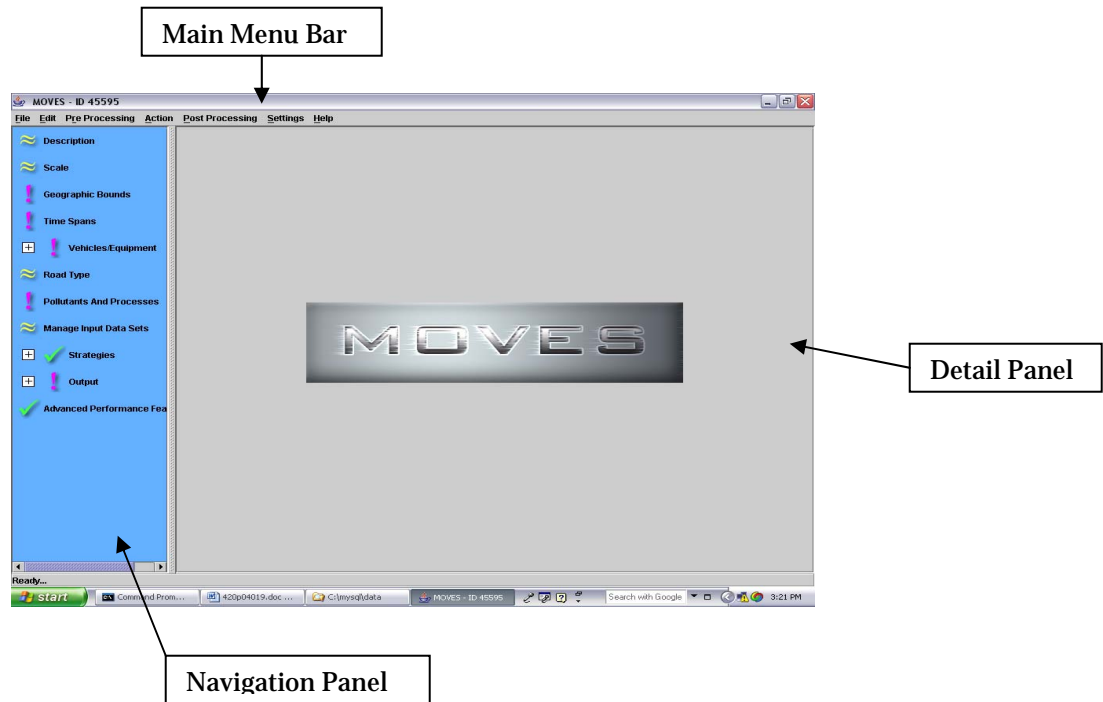
During execution of a RunSpec, a progress indicator will appear on the screen.

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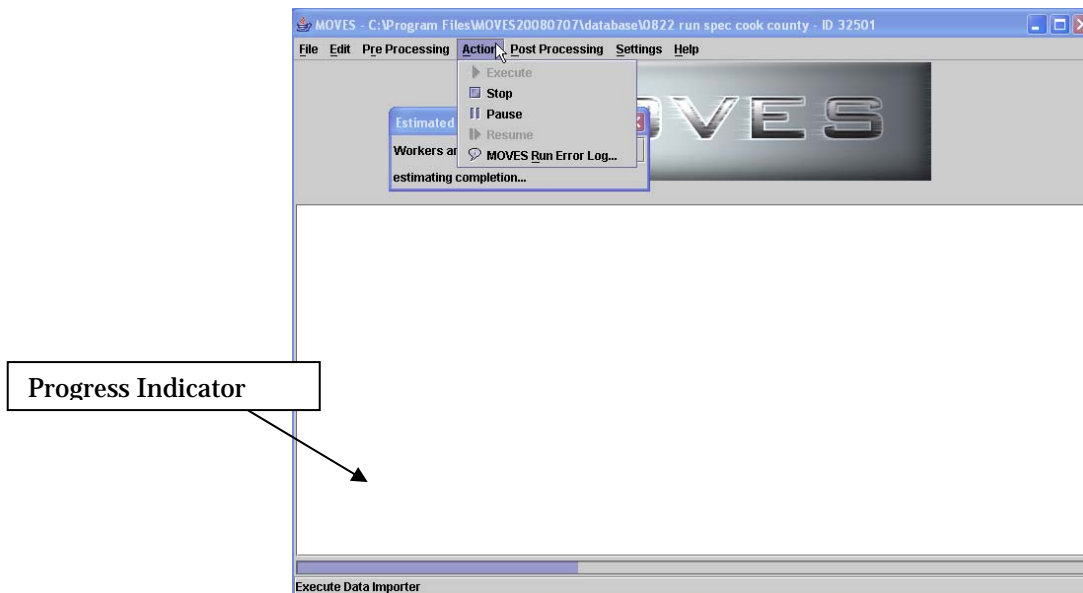
The Main Menu Bar will remain available, allowing the user to pause, resume, or stop (cancel) the RunSpec during execution.

The DRAFT MOVES2009 model contains a "mouse hover" user help feature that is present throughout the model. To utilize it, the user simply places the mouse over a particular MOVES GUI icon or command and keeps it very still. A small help box will appear containing

a brief text message; for example, the text message will contain 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 parameter icon 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.
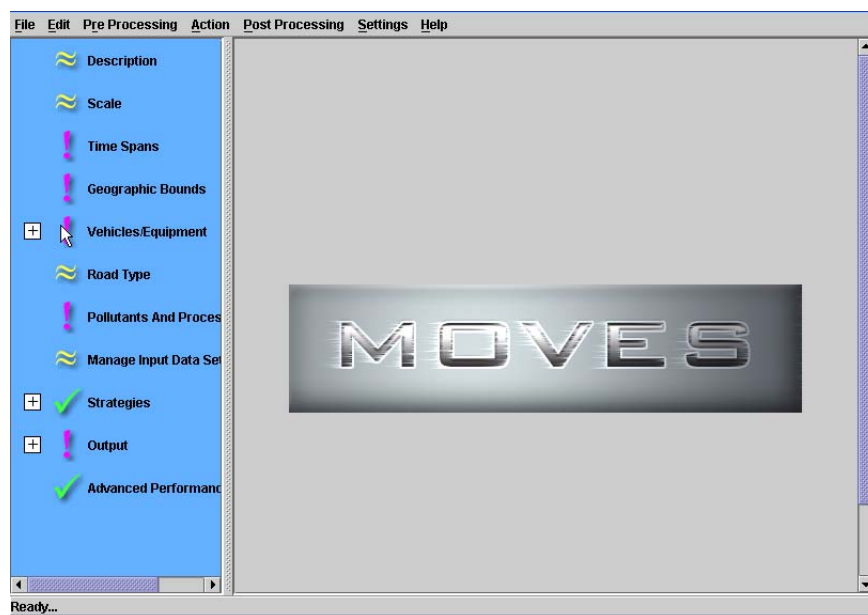
Selecting an item from the Navigation Panel will open a user interface in the Detail Panel in the center of the MOVES screen. In some cases, the RunSpec parameter listed on the Navigation Panel is broken down into sub-topics; these items are shown with an icon ($\boxplus$ or $\boxminus$) that allows 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.
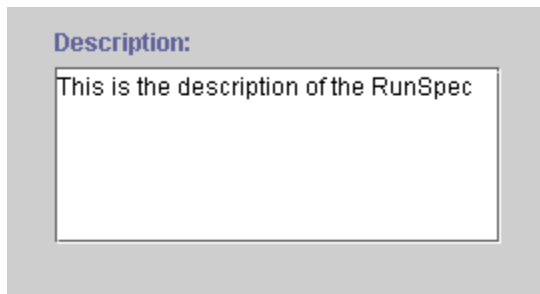
⚲ Note Parameters may be visited in any order. Selecting a parameter has no lasting effects until the RunSpec is saved or the model is executed. However, some parameters are linked (e.g. selecting 'County' or 'National' on the Domain/**Scale** panel result in different options on the **Geographic Bounds** user interface), so pay attention to the Navigation Panel icons after making a 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.

Description:

This is the description of the RunSpec

Up to 5,000 characters of text may be entered to describe the RunSpec. In the above example "This is the description of the RunSpec" is the text. Text entered has no effect on the results from the MOVES program.

!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 DRAFT MOVES2009 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.

If the user wants MOVES to use 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 2.3.3.2) 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, extended idling and evaporative 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 off-network common area). All of the project level data is entered in the Project Level Data Manager (see Section **2.3.3.3.4** 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**   DRAFT MOVES2009 is not capable of modeling any evaporative emission processes when using **Project Domain/Scale**. EPA plans to remedy this feature in future versions of MOVES.

       Only the Inventory Calculation Type may be selected in conjunction with the Project Domain/Scale. Selection of the Inventory produces an emission total over the selected domain (i.e., tons of CO per month) for either the entire project domain or an individual link within the project.

       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. The rate at which emissions occur (the mass and/or energy per unit of activity) is calculated if the users selects the **Emission Rates** button.

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

    1) the **Project Domain/Scale** cannot be chosen;

    2a) if **National Domain/Scale** is selected, only **"Zone & Link"** can be chosen in the "**Region**" section of the **Geographic Bounds** panel, or 2b) if **County Domain/Scale** is selected, only "**Zone & Link**" or "**Custom Domain**" can be chosen;

3) "**Off-Network**" is no longer listed as an "**Available Road Type**" on the **Road Type** panel;

4a) "**Start Exhaust**", "**Crankcase Start Exhaust**", "**Refueling Displacement Vapor Loss**", "**Refueling Spillage Loss**", "**Extended Idle Exhaust**", and "**Well-to-Pump**" emissions cannot be selected on the **Pollutants and Processes** panel; 4b) the "**Distance Traveled**" box on the **Pollutant and Processes** panel is automatically checked; 4c) a "**Running Exhaust**" process must be selected so that "**Distance Units**" can be chosen on the **General Output** panel; and

5) "**Road Type**" is automatically selected in the "**Road**" section of 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 segregated 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 be 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 policy guidance.

Caution!   Because of non-linearity in the general MOVES algorithm, some 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 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 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 "**Select All**" button to select all of the months. Similarly, the "**Clear All**" button removes all of the previously chosen months.

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.

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). Click one or more individual hours of the day to model 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 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 that are available on this panel are dependent on the choice of **Scale** (see Section 2.2.2 of this User Guide). If the user has chosen the **National Domain/Scale**, the following screen will appear:

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

$\mathscr{N}$**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.

$\mathscr{N}$**Caution!** Be advised that if an entire state or nation are run at the county level, execution times will 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.
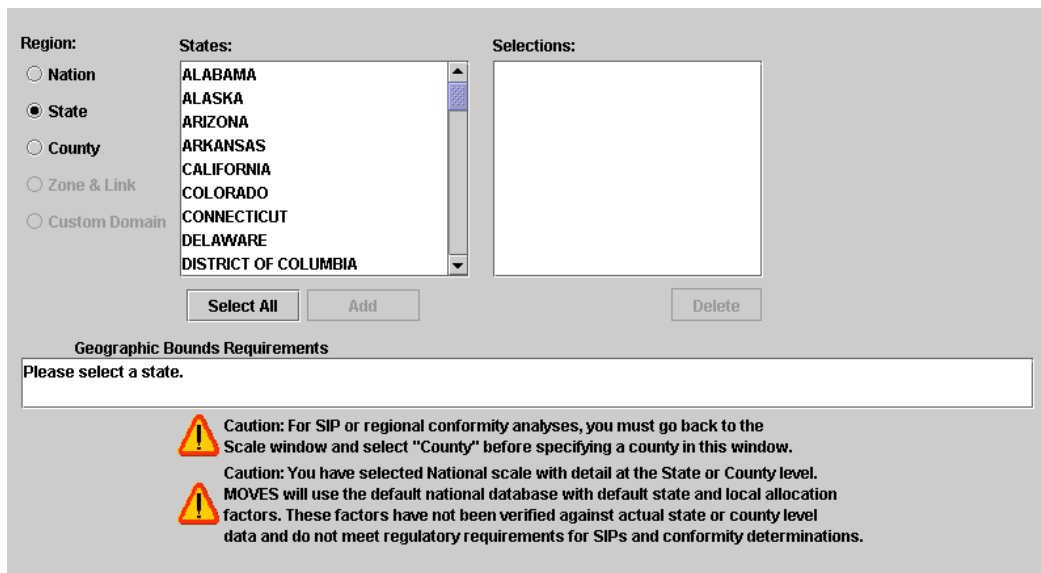
State: Choose "**State**" to run scenarios and compute emission inventories for particular States or Territories within the United States. This panel has two parts (**States,** and **Selections**) and buttons for choosing the appropriate selections.

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.

3. Highlight a previously selected state in Selections box. Click the "Delete" button to remove the highlighted state.

⚲ 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, which is outlined above.

If the user has defined the **Scale** of the model to be **County Domain/Scale,** the **Geographic Bounds** panel will contain an area in which to define the data source for the model, as shown in the following screenshot.

When using the **County Domain/Scale**, 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. The user can also define a "generic county" by selecting the **Custom Domain** option in the **Region** portion of the panel, as shown in the screen shot below.

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, altitude 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.

ℙ **Note** Altitude must be indicated by the user for two reasons: 1) older vehicles (without computer-controlled air/fuel ratios) tend to have higher emissions at high altitude compared to low altitude; 2) barometric pressure is a required input for some MOVES calculations, and average "Low" and "High" pressures are used depending on the user's selection. Future versions of MOVES will likely allow users to input local barometric pressure for custom domain rather than rely on these averages.

!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.

For both the **County** and **Custom Domain option**, the user must specify the user-supplied database that will contain county-specific data. 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**. For more information on how to import data using the **County Data Manager**, see section **2.3.3.3 Data Importer, County Data Manager and Project Domain Manager**.

!Tip The user should be sure to fill out all fields in defining the RunSpec before using the **County Data Manager**, as a filter is used when exporting default data that depends on selections the user makes when defining the RunSpec. Only the county, year, months, vehicles, road types, etc. that have been selected are exported. Some data may not be exported if the user selects the "**Export Default Data**" button before defining the Run Spec.

✐**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 scroll down lists (**Fuels** and **Source Use Types**) appear to the left of this panel to allow distinct selections for the vehicles/equipment.

Fuels:	Source Use Types:	Selections:
Compressed Natural Gas (CNG)	Combination Long-haul Truck	Gasoline - Passenger Car
Diesel Fuel	Combination Short-haul Truck	
Electricity	Intercity Bus	
Ethanol (E85)	Light Commercial Truck	
Gasoline	Motor Home	
Liquid Propane Gas (LPG)	Motorcycle	
	Passenger Car	
	Passenger Truck	
	Refuse Truck	
	School Bus	
	Single Unit Long-haul Truck	
	Single Unit Short-haul Truck	
	Transit Bus	

Select All   Select All   Delete

Add Fuel/Type Combinations

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.

4. Click "Delete" to clear highlighted selections from Selections box.

☟ **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 (e.g. Gasoline Intercity Bus), the subpanel will notify the user that the combination is invalid. These non-existent combinations do not need to be removed from the RunSpec, but no results are reported for such vehicles.

✐ **Caution!** The draft version of MOVES2009 lacks emission rates for motorcycles for most pollutants. Only energy consumption, greenhouse gas emissions and activity results will be generated for motorcycles. Criteria pollutant and air toxics emission factors for motorcycles are currently modeled as zero.

2.2.6 Road Type

If any on-road vehicle has been selected (see 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 scroll **down list.**
2. Click "**Add**" button to move choice to the **Selected Road Types** box.
3. Click "**Delete**" button to clear the highlighted selections.
4. Click "**Select All**" button to select all **Road Types** in the scroll down list.

⚲ Note The scroll down 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.

⚲ Note "**Off Network**" is automatically selected when start or extended idle processes are selected on the "selecting pollutants and processes" panel.

✎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..

		Running Exhaust	Start Exhaust	Brakewear	Tirewear	Evap Permeation	Evap Fuel Vapor Venting	Evap Fuel Leaks	Crankcase
☐	Total Gaseous Hydrocarbons	☐	☐			☐	☐	☐	
☐	Non-Methane Hydrocarbons	☐	☐			☐	☐	☐	
☐	Non-Methane Organic Gases	☐	☐			☐	☐	☐	
☐	Total Organic Gases	☐	☐			☐	☐	☐	
☐	Volatile Organic Compounds	☐	☐			☐	☐	☐	
☐	Carbon Monoxide (CO)	☐	☐						
☐	Oxides of Nitrogen	☐	☐						
☐	Primary Exhaust PM10 - Total	☐	☐						
☐	Primary PM10 - Organic Carbon	☐	☐						
☐	Primary PM10 - Elemental Carbon	☐	☐						
☐	Primary PM10 - Sulfate Particulate	☐	☐						
☐	Primary PM10 - Brakewear Particulate								
☐	Primary PM10 - Tirewear Particulate								
☐	Primary Exhaust PM2.5 - Total	☐	☐						
☐	Primary PM2.5 - Organic Carbon	☐	☐						
☐	Primary PM2.5 - Elemental Carbon	☐	☐						
☐	Primary PM2.5 - Sulfate Particulate	☐	☐						
☐	Primary PM2.5 - Brakewear Particulate			☐					
☐	Primary PM2.5 - Tirewear Particulate				☐				
☐	Total Energy Consumption	☐	☐						
☐	Petroleum Energy Consumption	☐	☐						
☐	Fossil Fuel Energy Consumption	☐	☐						
☐	Methane (CH4)	☐	☐			☐	☐	☐	
☐	Nitrous Oxide (N2O)	☐	☐						
☐	Atmospheric CO2	☐	☐						
☐	CO2 Equivalent	☐	☐						
☐	Benzene	☐	☐			☐	☐	☐	
☐	Ethanol	☐	☐			☐	☐	☐	
☐	MTBE	☐	☐			☐	☐	☐	
☐	Naphthalene	☐	☐			☐	☐	☐	
☐	1,3-Butadiene	☐	☐						
☐	Formaldehyde	☐	☐						
☐	Acetaldehyde	☐	☐						
☐	Acrolein	☐	☐						

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 that have been selected.

ꙶ **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 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 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.

If the user selects the "**Distance Traveled"** box, the MOVES model will compute and report the distance traveled by vehicles in the selected geographic, roadway, or time strata.

✿ **Note**   The "**Distance Traveled**" box will be 'grayed out' if "**Running Exhaust Process**" is not chosen in the Pollutants and Process selection matrix. The "Distance Traveled" calculation feature also requires that the user make a selection for "Distance Units" on the "General Output" panel; a reminder appears in the Pollutant/Process Requirements box as displayed in the screenshot below.
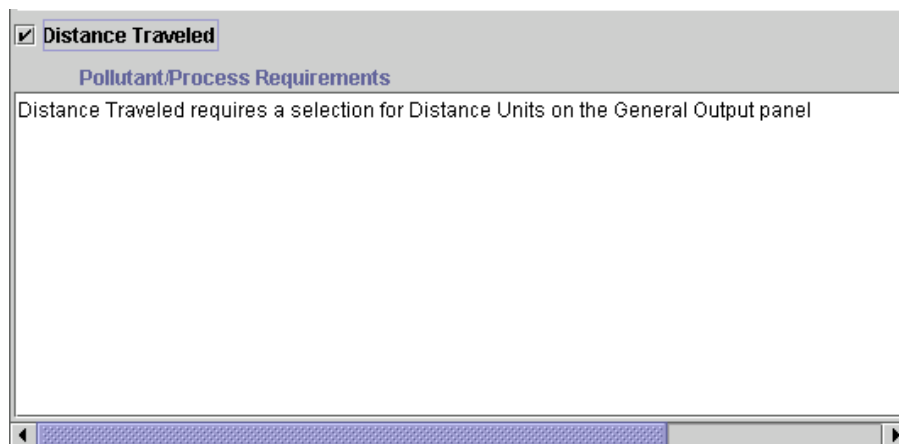
2.2.7.1 Processes

In DRAFT MOVES2009, "**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** (e.g., a combination long-haul truck idling overnight). 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

DRAFT MOVES2009 allows the user to calculate emissions of greenhouse gases, criteria pollutants, and air toxics associated with motor vehicle operation. MOVES also calculates energy consumption. For many pollutants, the emissions calculation in MOVES 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 the table below.

Pollutant Name	Abbreviation	Includes	Depends on
Total Gaseous Hydrocarbons	THC		
Non-methane Hydrocarbons	NMHC		THC and CH_4
Non-methane Organic Gases	NMOG		NMHC
Total Organic Gases	TOG		NMOG and CH_4
Volatile Organic Compounds	VOC		NMHC
Carbon Monoxide	CO		
Oxides of Nitrogen	NO_x		
Primary PM10 Total	PM10	PM10 OC, PM10 EC, PM10 SO_x	
Primary PM10 Organic Carbon	PM10 OC		PM2.5 OC
Primary PM10 Elemental Carbon	PM10 EC		PM 2.5 EC
Primary PM10 Sulfate Particulate	PM10 SO_x		Total Energy Consumption
Primary PM10 Brakewear Particulate			
Primary PM10 Tirewear Particulate			
Primary PM2.5 Total		PM2.5 OC, PM2.5 EC, PM 2.5 SO_x	
Primary PM2.5 Organic Carbon	PM2.5 OC		
Primary PM2.5 Elemental Carbon	PM2.5 EC		
Primary PM2.5 Sulfate Particulate	PM 2.5 SO_x		Total Energy Consumption
Primary PM2.5 Brakewear Particulate			
Primary PM2.5 Tirewear Particulate			
Total Energy Consumption	TEC		
Fossil Fuel Energy Consumption	FFEC		Total Energy Consumption
Petroleum Energy Consumption	PEC		Total Energy Consumption
Methane (CH_4)	CH_4		
Nitrous Oxide (N_2O)	N_2O		
Atmospheric Carbon Dioxide (CO_2)	CO_2		Total Energy Consumption
CO_2 equivalent	CO_2e	CO_2, CH_4, N_2O	
Benzene			VOC
Ethanol			VOC
MBTE			VOC
Naphthalene			PM10 Total, THC
1,3 Butadiene			VOC
Formaldehyde			VOC
Acetaldehyde			VOC
Acrolein			VOC

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 <u>must</u> 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.

1. 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.

2. 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**, Project Domain Manager and the **I/M Program Coverage Records Editor** 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** and **Appendix A: Displaying and Editing Vehicle Inspection and Maintenance (I/M) Program Coverage Information.**

!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.

3. Select the desired database from the dropdown list.

4. 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.

⚲ Note   The example in the figure above shows the addition of two alternate databases called "**UserInputDB1**" and "**UserInputDB2**" being added as MOVES inputs. The unspecified **Server** name implies that the databases reside in the MySQL Data folder on the user's local computer

!Tip The data in user input databases will "overlay," that is, augment or substitute for, data from the MOVES default database for the duration of the model run. Entries in the **Manage Input Data Sets** panel will not overwrite or alter the values in the MOVES default database.

5. Click the "**Move Up**" or "**Move Down**" buttons to modify the order in which a particular database selection is used.

!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.

6. Click the "**Delete**" button to delete selections.

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.   Draft MOVES2009 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 alternative fueled vehicles and advanced technology vehicles 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.

⚲ Note The AVFT and On-Road Retrofit interfaces were created early in the MOVES design process.   We hope to update these interfaces soon to be more consistent with the newer data importers available under the "**Pre Processing**" menu.

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 strategy data set name will appear in the window below "**Loaded objects**:". 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)".

Clicking the "**Import**" button allows the user to import an appropriate input file. The AVFT Strategy allows the user to import XML or tab-delimited text files. The On-Road Retrofit

Strategy allows the user to import Microsoft Excel (.xls), tab-delimited text files, or comma-delimited text. Typically, the import file is created by exporting a template file, editing it in a spreadsheet, saving the edited version as a tab-delimited text file, and importing this text file.

 ⚱ **Note** The AVFT will not import comma-separated data files.

2.2.9.2 Deleting Strategy Files

The user may delete a strategy data set from the RunSpec by clicking the "**Delete**" button. MOVES will prompt the user for confirmation before actually removing the data set.

2.2.9.3 Exporting Strategy Files

The "**Export**" button allows the user to write strategy parameter sets to file for storage. When the user clicks "**Export**", a standard Windows file browser appears from which the user can name a file to which to write the data as tab-delimited text.

To create a template file for the AVFT, the user should export a "default" set of parameters. To create a template file for the On-Road Retrofit strategies, the user chooses **"New,"** and then export a file containing only headers.   The AVFT also allows users to export parameters that were modified using the GUI. Both the AVFT and the On-Road Retrofit Strategies allow the user to export previously imported files.

2.2.9.4 Editing Strategy Files

The description of a strategy data set is displayed in the list of loaded objects and is set by selecting the object in the list and clicking the "**Description**" button. A dialog requesting the description is displayed:
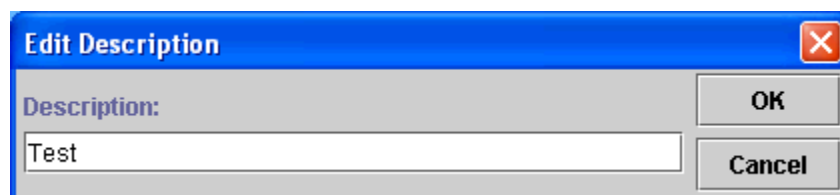
Because the description field can never be left completely blank, "*(default)"* is displayed automatically. This default description does not necessarily mean that the object contains only default data.

Although it is generally easier to export a file template and edit it outside MOVES, the **AVFT** strategy provides a GUI for editing the expected fractions of vehicle fuels and technologies by model year. This GUI is displayed to the right of the Strategies Management Panel. Users can edit data directly on the screen, as described below. The "**Cancel**" button undoes all edits the user has made. On-Road Retrofit parameters must be edited outside MOVES and then imported.

⁄**Caution!** Clicking the "**Cancel**" button will undo any edits that have been made within a strategy's GUI since the strategy was last created, loaded, or imported. Users should be cautious, as this can result in the loss of a significant amount of data. Periodic exports of the data, providing finer-grain control of rollback points, is recommended when a long data entry session is to be performed.

The "**Check**" button can be used to look for errors in the data inputs to a strategy. Clicking this button will display a popup message that either states that the data passed basic error testing, or explains why the data are not ready to be loaded into a RunSpec.

2.2.9.5 Saving and Transferring Strategy Files

Strategies can exist both inside and outside of a RunSpec, that is, the parameters can be saved as part of a RunSpec and/or as a separate file that can be re-used from one RunSpec to the next. This feature can help a user create a set of RunSpecs that use a consistent set of strategy settings. The "**Import**" and "**Export**" buttons allow for the transfer of strategy data between RunSpecs.

Using the **File** menu to save and open RunSpecs automatically saves and loads the **Strategies** embedded within the RunSpecs. The files containing the strategies do not need to be imported separately.

2.2.9.6 Varying Vehicle Fuels and Technologies using the AVFT Strategy

To vary the expected fractions of vehicle fuels and technologies by model year, the user should click on "**Alternate Vehicle Fuels & Technologies**" under Strategies on the Navigation Panel. This will open the **AVFT** screen.

The **AVFT** strategy allows the user to input alternative values for the FuelEngFraction MOVES database table. This table stores the fractions of engine and fuel technologies present within each model year for each source type. The AVFT also applies the default distribution of regulatory classes and vehicle size and weight bins to the "new" vehicles. For more information, see the Software Reference and Development Manual.

⚲ **Note** The AVFT does not modify the mix of cars and trucks in the fleet. This can be done by editing the SourceTypePopulation and VehicleTypeVMT tables using the Data Importer or County Data Manager.

On the **AVFT** panel, the user can select the desired source type from the drop-down list under "**Source Type**:". The table will fill with the data available for that source type and will display the allowed alternative fuels and technologies for that sourcetype.

Each column represents a combination of a fuel and engine type or a summary of several related vehicle types. Summaries are represented by gray columns and cannot be edited. The user can click in the header near the ">>" symbols to expand the summary columns to view their details.

	Gasoline Conventional Internal Combustion	Diesel Fuel Conventional Internal Combustion	Advanced Gasoline <<	Gasoline Advanced Internal Combustion	Gasoline Conventional Internal Combustion - Hybrid - Mild	Gasoline Conventional Internal Combustion - Hybrid - Full	Gasoline Advanced Internal Combustion Hybrid Mild	Gasoline Advanced Internal Combustion Hybrid Full	Advanced Diesel >>
2000	99.70%	0.08%	0.18%	0.03%	0.08%	0.08%	0.00%	0.00%	0.0
2001	99.70%	0.08%	0.18%	0.03%	0.08%	0.08%	0.00%	0.00%	0.0
2002	99.51%	0.07%	0.38%	0.14%	0.12%	0.12%	0.00%	0.00%	0.
2003	99.10%	0.10%	0.77%	0.27%	0.25%	0.25%	0.00%	0.00%	

Source Type: 21 Passenger Car   Normalize   Add Model Year

.

The user can click the "<<" symbols on the expanded columns to collapse them to summary-only display. The user can change any non-gray number by clicking on it. All numbers are entered as percentages, not fractions. To enter the value of 25% (i.e., fraction 0.25), the user can click and type either "25" or "25%". Both are equivalent. Typing 0.25 will be interpreted as 0.25% (i.e., fraction 0.0025).

⚲ **Note** Unique energy consumption and GHG emission rates are available for most fuels and technologies. However, in Draft MOVES2009, the criteria pollutant and air toxics rates for many alternative fuels and technologies have been populated with emission rates identical to those for conventional fuels and technologies.

2.2.9.7 Normalizing Data

All entries in a single row must sum to exactly 100%. The "**Normalize**" button can be used to adjust ratios so that all add up to 100%. For instance, if two cells are populated on a model-year line, entering 1% and 4% and then normalizing would change the percentages to 20% and 80%. This maintains the original 1:4 ratio while still totaling 100%. The **AVFT** data cannot be exported, nor can the simulation run be started, until all model-year rows on all source use types meet the above rule. Clicking the "**Normalize**" button is the easiest way to ensure that this condition is met.

2.2.9.8 Adding Model Years

The **AVFT** panel is initialized with the contents of the MOVES Default database FuelEngFraction table, including only those model years present in the database. The "**Add Model Year**" button can be used to add model years in addition to those displayed. Clicking this button will copy the data from the highest model year for all source use types to the next model year.

⚲ **Note** The **AVFT** strategy only works with model years from 2001 to 2050, inclusive, and will generate an error message denying the action if an attempt is made to work outside of these bounds. If all years from 2001 to 2050 are displayed, the "**Add Model Year**"

button will be grayed out. The AVFT was not designed to work for model years prior to 2001.

2.2.9.9 Editing AVFT files outside of MOVES

To create AVFT strategy files outside of MOVES, it is easiest to start by exporting the default parameters as a template, then opening the saved file with a spreadsheet program such as Microsoft Excel.   The first cell will be the description, which the user should edit to describe the new AVFT.

The second row contains field names: "SourceTypeID", "Year", "Category", "fueltypeID", "engTechID" and "Amount".   SourceTypeIDs and FuelTypeIDs are defined in the output section of the MOVES Software Development and Reference Manual. "Year" refers to the model year of the vehicles.   "Category" refers to the grouping in the AVFT display. "EngTechID" refers to the engine technology type, described below.   "Amount" refers to the fraction of the sourcetype in that model year that have that combination of fueltype and engine technology.   Not all fueltypes and engine technologies are relevant for all sourcetypes.

Engine Technologies

engTechID	EngineTechnology Name
1	Conventional Internal Combustion
2	Advanced Internal Combustion
11	Moderate Hybrid - Conventional Internal Combustion
12	Full Hybrid - Conventional Internal Combustion
20	Hybrid - Advanced Internal Combustion
21	Moderate Hybrid - Advanced Internal Combustion
22	Full Hybrid - Advanced Internal Combustion
30	Electric
40	Fuel Cell
50	Hybrid - Fuel Cell

Beneath the field names are data rows. For Draft MOVES2009, there are about 7050 rows in an AVFT file. Each of the rows in the default AVFT should be retained.   The user should edit only the "Amount" in each row. For each combination of sourcetype and model year, the sum of the "Amount" should equal 1.

The edited AVFT file should be saved as a tab-delimited text file and then imported into the desired RunSpec(s) using the "Import" button.

2.2.9.10 Managing On-Road Vehicle Retrofit Data

DRAFT MOVES2009 has the capability to model on-road vehicle retrofit strategies for all exhaust pollutants for diesel trucks and buses. To add on-road vehicle retrofit data, the user should click the **On-Road Retrofit** option under **Strategies** on the Navigation Panel. This will open the **On-Road Retrofit** panel, which displays the **Strategies Management** Panel on the left side of the panel as described earlier in this section. The "**Description**", "**Cancel**", "**New**", **"Export",** "**Delete**", and "**Check**" buttons work as described earlier in this section. Appendix D provides detailed instructions for creating an **On-Road Retrofit** data file.

⚓ Note The Retrofit Strategy only works for non-passenger diesel trucks and buses. Importing a strategy for motorcycles, cars, passenger trucks or for other fuel types will generate an error.

The "**Import**" button differs from the AVFT import as is described below.
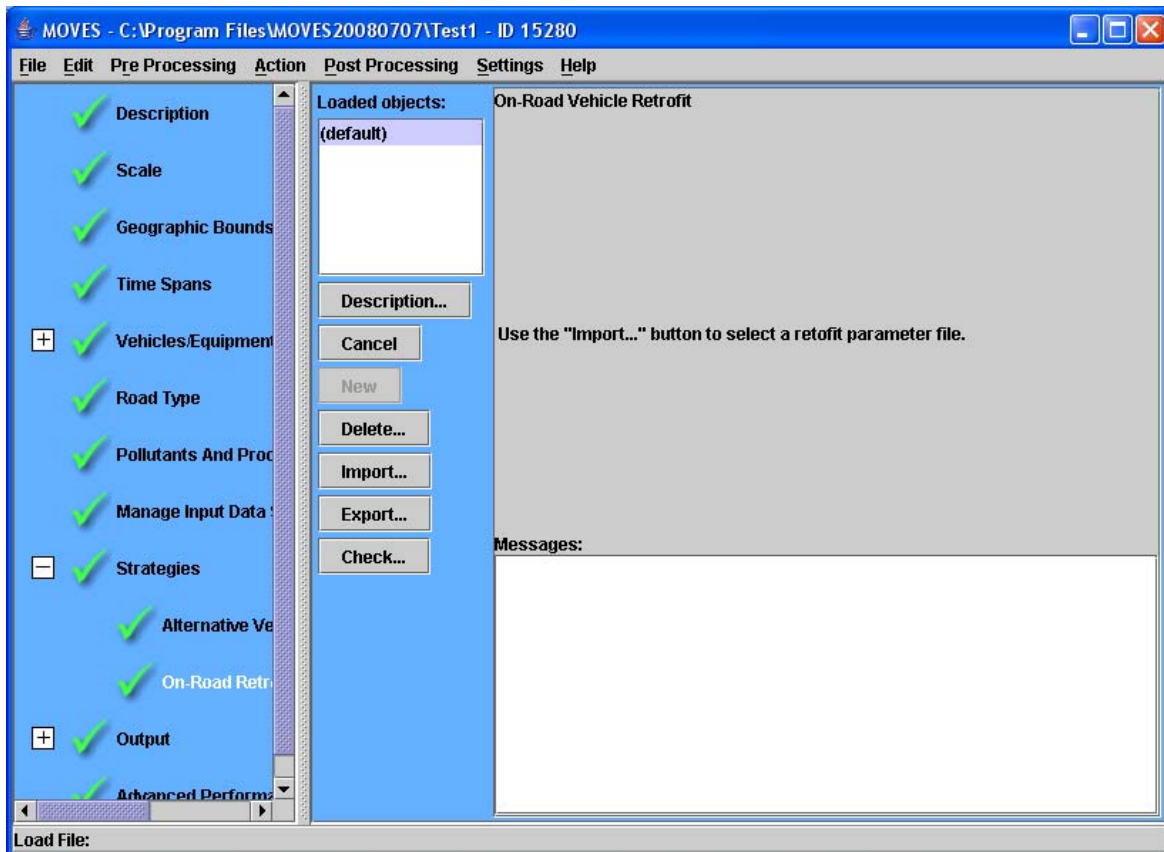
2.2.9.11 Importing Retrofit Data

When the user clicks on "**On-Road Retrofit**", the Detail Panel to the right of the **Strategies Management Panel** instructs the user to click the "New" button first and then click the "**Import**" button to select a retrofit parameter file. The "**Import**" button enables the user to import On-Road Retrofit data as a tab delimited, comma separated (.csv), or .xls file. When the user clicks on "**Import**", a standard Windows file browser window appears from which the user can select the file. If an .xls file is imported, MOVES will prompt the user to specify a worksheet from which the data should be imported

Once data are imported, the display in the Detail Panel will show the name and location of the imported file ("Data Source"). A "**Reload**" button that will also appear next to the file name can be clicked on to capture any changes to the imported data file.

Importing retrofit data automatically saves the data to the RunSpec, but it does not automatically apply the retrofit parameters to the emission results that MOVES produces. To apply the retrofit parameters to the emission results, the user <u>must</u> check, the **Use the On-Road Retrofit** box that appears at the top of the On-Road Retrofit panel. This allows the user to turn retrofit on and off without reloading the files.

If there are any data format or consistency issues in a file that the user is trying to import (e.g., incorrect number of columns or data that are inconsistent with what the user selected elsewhere in the MOVES GUI), a message(s) will appear in the **Messages** window at the bottom of the panel.

2.2.10 Output

Selecting **Output** on the Navigation Panel expands the list to show **General Output** and **Output Emissions Detail**. Each of these allows the user to specify aspects of the output data.

2.2.10.1 Specifying Database, Time Period, and Units in Output

Click on the **General Output** panel to specify the output database, the time period, and the units. The panel has two sub-panels: **Output Database** and **Output** (showing units).

2.2.10.1.1 Output Database

The first box, **Output Database**, contains input boxes for the server name and the output database name. A blank server box is the default and it indicates that the localhost (the MySQL database located on the user's computer) is to be used. Most users should leave this blank. The second box contains the name of the MySQL output database where the user desires the MOVES output to be written. The user must select an existing MOVES output database or create a new MOVES output database into which the results should be placed. If the output database already exists and contains rows within its output tables, a warning icon is displayed. A new run of the MOVES model will add its results to an existing MOVES output database, and automatically

give them the next available MOVES Run identification number. MOVES output records in an existing output database cannot be deleted or replaced from within the MOVES GUI.

A user choosing to create a new MOVES output database must choose a unique name and may click the "**Create Database**" button. All new output databases begin with a MOVES Run identification number of one.

✎**Caution!** The user should <u>never</u> attempt to create a new output database with the same name as the MOVES default database or of any user-supplied MOVES input databases.

2.2.10.1.2 Output

Output is the second box. It contains user choices for the mass, energy, and distance units in which the results will be reported. The time units are set to correspond to the time reporting level (see next section) and are displayed, but they are not user-selectable on this panel. The possible choices for mass are kilograms, grams, pounds, or US tons (short tons). The possible energy units are Joules, Kilojoules, or Million BTUs (British Thermal Units). The possible distance units are miles or kilometers. Distance units may be selected only if "distance" output has been requested on the pollutants and processes panel. The user may choose only one unit for each of the four types. The units are reported in the output database in the MySQL output table "movesrun."

!Tip Make sure to choose units of the appropriate magnitude for your RunSpec. For instance, if the RunSpec is very limited spatially, temporally, and in terms of vehicle and road types, then choosing "**Tons**" as the "**Mass Units**" will not give you the desired level of detail. Also, note that MOVES might round the results down to zero.

The checked box **Time Factors** (Mass per time unit/Energy per time unit) in the **Output** sub-panel reminds users that all MOVES output is for a certain period of time. The "**Time Units**" is always set equal to the time reporting level, so the MOVES Inventory output results are reported as the total mass of emissions or total energy over the output time period defined by the RunSpec.

2.2.10.2 Specifying Emission Distinctions in Output

Select **Output Emissions Detail** on the Navigation Panel (after expanding **Output**) to specify distinctions desired in the output data. The **Output Emissions** Detail Panel consists of four sub-panels: **Always**, **for All Vehicle/Equipment Categories**, **Estimate Uncertainty**, and **On Road/Off Road**.

⚲ **Note** Users may eliminate unneeded output details using this screen or using post-processing.
Selecting detailed output here and using MySQL command to aggregate the output to the
desired level may be preferable since this approach allows the user the option of retrieving
the more detailed output without re-running the model. However, if a run includes many
times, locations, sourcetypes, etc., the user may want to reduce the number of items
selected on this screen to avoid producing unmanageably large output files.

2.2.10.2.1 Always

The **Always** box at the upper left is a reminder that the output data will always contain
dimensions for time, location, and pollutant. The **Always** box also contains two buttons that
allow the user to specify the level of output aggregation by time and location level.

By default, the "**Time**" button will report the same time level as selected in the **Time
Spans** panel. The user may change the time level in the "**Time**" button to a value that is equal or
longer than the **Time Aggregate Level** specified in the **Time Spans** panel. For example, if the
Time Aggregate Level is set to "**Hour**" in the **Time Spans** panel, then the "**Time**" button can be
set to "**Hour**", "**24-Hour Day**", "**Portion of Week**", "**Month**", or "**Year**".

The "**Portion of Week**" selection requires explanation. The "24-Hour **Day**" selection
does not select a particular day of the week but a *kind of day of the week* and the results are for
one day of each kind of day requested in the RunSpec (e.g. one weekend day and/or one
weekday). In the default case where the MOVES input database specifies two kinds of days,
reporting by "**Portion of Week**" reports weekday results as a total of five weekdays and
weekend results as a total of two weekend days. See the MOVES Software Design Reference
Manual for more information.

By default, the "**Location**" button will report the same location level as selected in the
Geographic Bounds panel. The user may change the location level in the "**Location**" button to
a value that is equal to or larger than the level set in the **Geographic Bounds** panel. For

example, if the **Geographic Bounds** panel is set to "**County**", then the "**Location**" button can be set to "**County**", "**State**", or "**Nation**".

⚲ **Note**   The choice of "**State**" or "**Nation**" will aggregate results from the areas chosen in the **Geographic Bounds** panel. For example, if only two counties from a state are specified in a RunSpec, the output results for "**State**" will be the sum of the emissions from only these two counties.

A "**Pollutant**" button is not available because pollutants are always reported separately in the output (i.e., there is no aggregation across pollutants).

2.2.10.2.2 For All Vehicle/Equipment Categories

The dimensions that apply to both on road and off road sources are "**Model Year**", "**Fuel Type**", and "**Emission Process**". Checking any or all of these items means that the output will be distinguished by that factor. For example, if "**Emission Process**" is checked, the output will be distinguished by the emissions processes selected, such as Running Exhaust, Start Exhaust, etc.

2.2.10.2.3 On Road/Off Road

DRAFT MOVES2009 does not include the ability to model Off Road emissions. Thus, within the **On Road/Off Road** only the **On Road** section is functional. In the **On Road** box, options are available to report by "**Road Type**", "**Source Use Type**", or **Source Classification Code (SCC)**". If none of these options are checked, no vehicle or road classification distinctions will be made in the output. Instead, the results will be reported as summed across all vehicle and road classes.

!Tip "**Source Use Type**" and "**SCC**" are mutually exclusive. **Source Use Types** are native to MOVES, so this is the preferred option. MOVES can report by Source Classification Code (SCC), but this introduces an additional approximation step in the calculations to convert output by **Source Use Type** to SCC.

Several interdependencies exist among these items. For example, reporting by SCC implies that FuelTypes, but not RoadTypes will be distinguished.

2.2.10.2.4 Estimate Uncertainty

The **Estimate Uncertainty** box should be left unchecked by most users. If checked, the remainder of the uncertainty panel becomes active for the user to specify a number of iterations, whether randomly sampled input used for each iteration should be reported, and whether the output produced from each iteration should be reported. This mechanism may be used to investigate the uncertainty of MOVES results due to uncertainty in some of its inputs. Users should consult the uncertainty section of the MOVES Software Design and Reference Manual before attempting to use this feature. The default database provided with DRAFT MOVES2009 does not contain realistic input data for this feature.

✎**Caution!** Model execution time and the volume of output produced (if the output of each iteration is saved) increase roughly in proportion to the number of iterations specified. If the randomly sampled inputs for each iteration are also saved, the resulting output will be even more voluminous.

2.2.11 Advanced Performance Features

This **Navigation Panel** item invokes features which may be used to diagnose problems with the model software, or to improve the run time for long model runs by saving and reusing intermediate results. Use of these features requires knowledge of the detailed software components of the MOVES program, the inputs they require, and the outputs they produce. Users interested in these features should review the MOVES Software Design and Reference Manual.

This panel allows users save intermediate results from various portions of the model and to turn off portions of the model code.

This panel also allows the user to "turn off" final aggregation of MOVES results. Selecting the "Do Not Perform Final Aggregation" option can reduce execution time for large

runs. However, when this option is selected, the MOVES output tables may include rows with duplicate key fields. The results from these rows should be summed to produce useful output.

And, this panel allows users to specify the database to be used when modeling a Custom Domain. See Section 2.2.4 Geographic Bounds for additional information on the Custom Domain option.

⫽**Caution**! It is possible to undo a custom database selection by hitting the "Refresh" button, which clears the selected database.

2.3 Main Menu Bar

The Main Menu Bar runs across the top of the screen and contains seven drop down menus: **File, Edit, Pre Processing, Action, Post Processing, Settings,** and **Help**. Menus can be opened by either clicking on the menu command, or by using a keyboard combination, defined as ALT and the underlined letter of the menu command (e.g., ALT + F for the **File** menu, ALT + R for the Pre Processing menu).

2.3.1 File

File provides a drop-down menu typical of those used when manipulating documents. Click **File** to open the drop down menu and then on a menu item to select **New, Open, Close, Save, Save As, Print, Exit**, or to choose to open recently opened files. Use the **File** commands to create, load, and save RunSpec objects. The Example RunSpec may be opened from this menu (see Section 1.4 for details). DRAFT MOVES2009 provides the option to execute file manipulation commands using keyboard combinations as well; for example, the **File Open** procedure can be initiated by selecting the **Alt** and **O** keys simultaneously when the File drop down box is opened. Key sequences required to execute the file manipulation commands are displayed using the "mouse hover" feature while placing the cursor over the desired command.
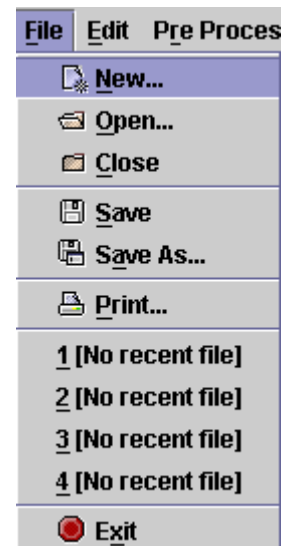
2.3.2 Edit

Edit provides the usual cut/copy/paste commands. These commands are only available in text entry sections of MOVES RunSpec. Keyboard combinations are available to execute edit commands when the Edit drop down box is opened, and can be identified using the "mouse hover" feature.

⫽**Caution!** MOVES keyboard combinations use different letters than Microsoft applications for the same functions; for example, the cut command in MOVES uses Alt and T, and paste uses Alt and P.

2.3.3 Pre Processing

Pre Processing contains four active menu options, **Display/Edit IM Program Coverage Records, Data Importer, County Data Manager**, and **Project Domain Manager**. A description of each active menu option follows:

2.3.3.1 Display/Edit I/M Program Coverage Records

The **Display/Edit I/M Program Coverage Record**s menu option allows the user to display and change the information about vehicle inspection maintenance programs that apply to the current run specification. Because using this feature requires an understanding of other parts of the RunSpec, it is covered in Appendix A.

2.3.3.2 Data Importer, County Data Manager and Project Domain Manager

The **Data Importer**, **County Data Manager** and **Project Domain Manager** menu options enable the user to create user-supplied databases for direct entry of data into MOVES.

The **County Data Manager** and the **Project Domain Manager** are means by which the user can define the county-specific or project-specific data required if the user selects **County Domain/Scale** or **Project Domain/Scale** when defining the **Scale** parameter of the RunSpec. Neither the **County Data Manager** nor the **Project Domain Manager** menu options will be enabled unless the user has made the **County Domain/Scale** or **Project Domain/Scale** choice.

The **Data Importer** allows the user to select individual importers "ala carte" from the importer menu. The **County Data Manager** and **Project Domain Manager** are designed to specify all the inputs required for certain kinds of runs and, thus, do not permit such "ala carte" selections. The **Project Domain Manager**, in particular, requires the user to provide virtually all of the data required to model a specific project.

📏**Caution!** All tabs/fields in the **County Data Manager** and **Project Domain Manager** must be completed to ensure the output generated is correct. In some cases, a RunSpec may execute even though all the tabs in the **County Data Manager** or **Project Domain Manager** were not completed; however, the results from such RunSpecs may populate the incomplete tabs with default data, which may not be appropriate due to how MOVES generates, calculates, and allocates various data.

!Tip The user does not have to specify a database containing county or project data that was created with the **County Data Manager** or **Project Domain Manager** in the **Manage Input Data Sets** panel. These databases only have to be selected in the **Domain Input Database** pane of the **Geographic Bounds** panel. However, a database created with the **Data Importer** should be input using the **Manage Input Data Sets** panel.

From the user perspective, there are two major differences between working with importers through the **Data Importer** and working with the same importers through the **County Data Manager** and the **Project Domain Manager**. First, as "domain managers" the **County Data Manager** and **Project Domain Manager** indicate whether the user has imported all of the data that is needed to establish a new domain. To this end, the **County Data Manager** and the **Project Domain Manager** will indicate by red "x-circles" where data that is needed on the importer tabs (see screenshot below) in order to establish the domain**.** Green "check-circles" indicate where data has been properly imported.
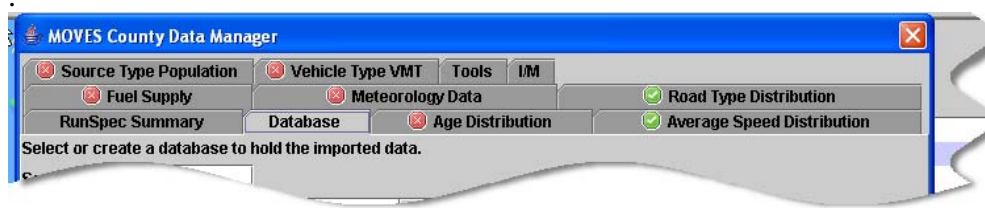
Secondly, because of the level of detail required and the structure of the underlying data tables, the **County Data Manager** restricts the RunSpec to a single county and single calendar year, and the **Project Domain Manager** restricts the RunSpec to a single hour, daytype, month and county.

This user guide will use the **Data Importer** screens to explain functionality of individual importers.   Specific instructions associated with using the **County Data Manager** or the **Project Domain Manager** to establish a new domain are included where appropriate.

When the user selects one of the data manager options, the **MOVES Data Importer** window will open. This window contains several tabs to aid the user in defining the data source, destination, and database structure for the imported data. **Importer** tabs provide templates for collecting user-provided data.

2.3.3.2.1 Database

The **Database** tab is the default tab for the **Data Importer** menu option. It contains a scrollable log detailing the success of the data import and other helpful diagnostics, and opens a window that allows the user to identify the database where the imported data will be stored. Data can not be imported until a receiving database is identified.
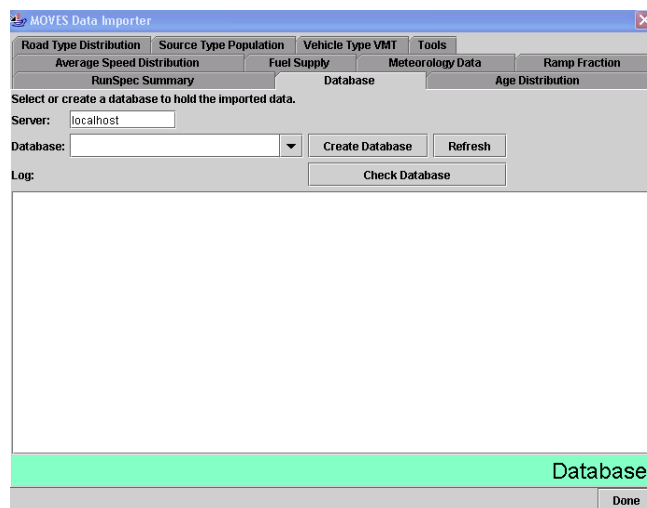
By default, the host server for the receiving database is "localhost" (generally the user's own machine). The user can specify a different host server by entering the name of the alternate server in the **Server** box. The receiving database can be identified by clicking on the drop-down arrow adjacent to the **Database** box and making a selection from the list. Alternatively, the user can create a new database by typing a database name in the **Database** box and clicking the

"**Create Database**" button. Clicking the "**Refresh**" button will clear the database selection from the **Database** box display and refresh the list of databases.

📍**Caution!** The user should <u>never</u> attempt to create a new input database with the same name as the MOVES default database or of any user-supplied MOVES output databases.

The Database tab also contains a **Log** panel. This panel is used by the system to display the results of the import process. Once the user chooses either the **Tools** or an **Importer** tab to import data into the selected database, the user can refer to the **Log** panel in the **Database** window to view the results of the import process.

2.3.3.2.2 RunSpec Summary

The **RunSpec Summary** tab opens a window which provides a summary of RunSpec parameters defined by the user. The user should choose this tab to see an overview of the RunSpec. In setting up the RunSpec, the user defines modeling parameters that limit the scope of the modeling process; templates and export files created with the **Data Importer** will be limited by choices the user makes in defining the RunSpec.
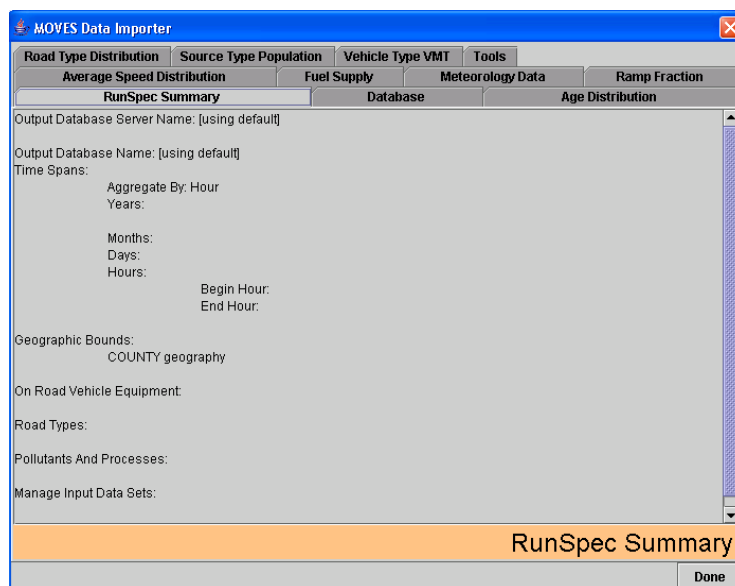
The **RunSpec Summary** window of the **Data Importer** displays the geographic, time span, road type, vehicle, pollutant and process, and input data set choices made by the user. The window is auto-generated and does not accept or require user input.

2.3.3.2.3 Tools

The **Tools** tab is provided to facilitate batch input of imported data into the MOVES system, such as for a series of data import files representing all the counties in a state. When the user clicks the "**Generate Importer XML File**" button after completing an import, the system will write an importer instruction file in Extensible Markup Language (XML), containing a statement that replicates the import done through the GUI.

Using an XML or text editing program, the user then can copy and edit the importer XML file to create a series of similar, but not identical import statements. The resulting XML files can be called from the command line, generating a series of user-input databases. For more information on running MOVES from the command line, see Appendix C.

2.3.3.2.4 Data Managers (Importers)

The remaining tabs in the **Data Importer** menu option are **Data Manager** or **Importer** tabs. These tabs contain templates and rules that aid the user in bringing specific types of data into the model. By choosing a **Manager** or **Importer**, the user gains access to several tools that will help create a successful data import.

!Tip The user should be sure to fill out all fields in defining the RunSpec before using the **County Data Manager** or the **Project Domain Manager**, as a filter is used when exporting default data that depends on selections the user makes when defining the RunSpec. Only the county, year, months, vehicles, road types, etc. that have been selected are exported. Some data may not be exported if the user selects the "**Export Default Data**" button before defining the Run Spec.

!Tip EPA is developing MOBILE6 converters to help users prepare import-ready data files from their existing MOBILE6 files.   When complete, these converters will be available on the MOVES website.

The user should select a **Manager** or **Importer** tab to choose a specific importer to assist with specialized data file structure definition and to complete the import process. Importers for the following tables are included in DRAFT MOVES2009:

MOVES Table	Data Importer	County Data Manager	Project Domain Manager
RampFraction	X	X	
RoadTypeDistribution	X	X	
SourceTypePopulation	X	X	
VehicleTypeVMT	X	X	
ZoneRoadActivity	X (Custom Domain Only)	X (Custom Domain Only)	
AgeDistribution	X	X	X
AverageSpeedDistribution	X	X	
FuelSupply	X	X	X
MeteorologyData	X	X	X
OperatingModeDistribution			X
LinkSourceType			X
LinkDriveSchedules			X
Off-Network			X

When the user selects one of these tabs, a panel opens that allows the user to define the import parameters. Each **Importer** tab is organized in the same way, with a place for the user's description of the data, a place to browse for data files, buttons to create templates, import, export and clear, and a space for messages.

The user can click the "**Create Template**" button to create an import template file and a set of reference files. After selecting the "**Create Template**" button, the user must define the file, file path and extension and click the **Save** button. The template and reference files will be specific to the RunSpec. The template will include required data field names and often will be populated with the required key fields. The reference files will provide lists and descriptions of the relevant MOVES id fields (i.e., countyID codes, fuelformulationID codes, roadtypeID codes, hourDayID codes, and sourcetypeID codes). The user can then edit the template with a spreadsheet application or other tool before importing the data files into DRAFT MOVES2009.

Although the meaning of many data element names may seem clear, the user should consult the reference tables provided with the template to determine appropriate content for the fields. For example, although fuelYearID might seem to refer to the calendar year, it actually refers to "fuelyear" and does not have a one-to-one mapping with calendar year.

In many cases, users can import duplicate information by using "wildcards" in the input files. For example, using the word "ALL" in a sourceTypeID column of the input file will result in records with the same information for each source type selected in the runspec.

✔Caution!   "ALL" produces inputs only for all the sourcetypes in the **RunSpec** rather than all the sourcetypes in the default database. For this reason, users are strongly urged to create a comprehensive runspec prior to using the importers.

The hourDayID column supports a special form of the "ALL" wildcard. It allows the user to get all 48 hour intervals (24 x 2 day types) or just the hours in a given day type. Remember, these are constrained by the runspec's day and hour selections.

The affected fields are:

Column	Wildcard
dayID	ALL
hourID	ALL
hourDayID	ALL ALL Weekday ALL 5 5* *5 ALL Weekend ALL 2 2* *2
monthID	ALL
yearID	ALL *Note: Only one year is allowed in the CDM.*
roadTypeID	ALL
sourceTypeID	ALL
hpmsVTypeID	ALL
countyID	ALL *Note: Only one county is allowed in the CDM.*
zoneID	ALL

In the receiving database, wildcard data is fully expanded.   Importing a file that uses wildcards and then exporting again will produce a file with the full data set.

Once a correctly formatted source file is created, the user should use the "**Browse**" button to find and open the source file. The file path and name of the selected import file will appear in the panel.

!Tip   Make sure to save any changes made to a source file before you import it.

       Once a file has been selected for import, the user may click the "**Import**" button to bring the data into MOVES via the file and path defined in the **Database** window. If the file contains multiple worksheets, the user is prompted to select the appropriate worksheet as well. The user should refer to the **Messages** box in the **Importer** tab or the **Log** panel in the **Database** window to determine the status of data import.

!Tip   See the **Messages** box for information on import problems and basic information for fixing problematic import files.

       Imported data can be cleared from the database by clicking the "**Clear Imported Data**" button. When the "**Clear Imported Data**" button is clicked, the message box shown below pops up.

Clicking "Yes" clears the data, while clicking "No" cancels the action and allows the user to export the imported data, as described below.

!Tip   Users should "**Clear Imported Data**" before reattempting to import a file that resulted in an error message.

       Data files can be exported to an Excel spreadsheet or text file using either the "**Export Default Data**" or "**Export Imported Data**" button; however, not all tabs in the **County Data Manager** have the "**Export Default Data**" button. Those without it are noted in the sections below. Also, some tabs in the **Project Domain Manager** include an additional export button, "**Export Most Recent Execution Data**". Use of this button requires a previous execution of the MOVES model in order to generate the data which is to be exported.

       Clicking on the "**Export**" button will cause the **Export [*Importer*] Data** window to open. The user defines the file and file path in the **Export Data** window, and clicks the "**Save**" button, to export the data to the specified location.

       Occasionally, clicking on one of the "**Export … Data**" buttons may result in the following pop-up window:

No Data

No data was retrieved. Consult the user guide for more options.

OK

This message simply means that there were no data available to export.

2.3.3.2.4.1 Fuel Supply Importer

The **Fuel Supply** importer allows the user to assign existing fuels to counties, months, and years, and to assign the associated market share for each fuel. The market share for a fueltype (gasoline, diesel, etc.) must sum to one for each county, fuelyear, and month. In Draft MOVES2009, the user must select fuels from those available in the default database. The importer does not allow a user to add new fuels.

This importer also allows the user to supply a data element, "marketShareCV", which is the coefficient of variation for the market share. This measures uncertainty associated with the market share determination, and is used if the user selects **"Estimate Uncertainty"** in the **"Output Emissions Detail"** part of the RunSpec. If the user does not select "**Estimate Uncertainty"**, the value is not required and may be left blank.

📏**Caution!** Although fuelYearID might seem to refer to the calendar year, the year table in the Draft MOVES2009 default database maps all years 2012 through 2050 to the "fuelyear" 2012; thus for years 2012 through 2050, the fuelyearid 2012 should be used.

!Tip   Modeling different fuels in different years (2012-and later) can be done with a separate run for each year assigning different fuels to the "2012" fuelyearid for each run.

2.3.3.2.4.2 Vehicle Type VMT

The **Vehicle Type VMT** importer allows the user to enter yearly vehicle miles traveled. These values will represent national values for the general **Data Importer** and county-year specific values for the **County Data Manager**. Yearly VMT is required regardless of the time span selected for the run.

This tab does not have an "**Export Default Data**" button and the user is required to acquire/generate this data indepently if the **County Domain/Scale** is chosen. Data must be supplied for each HPMSVtype (e.g., passenger cars, combination trucks, etc.) implied by the Source Use Types selected in the RunSpec.

2.3.3.2.4.3 Source Type Population Importer

The **Source Type Population Importer** allows the user to input the number of vehicles in the geographic area which is to be modeled for each vehicle or "source type" selected in the RunSpec. This manager does not allow input of either the sales growth rate or the migration factor.

This tab does not have the "**Export Default Data**" button and the user is required to acquire/generate this data independently if the **County Domain/Scale** is chosen. Data must be supplied for each source type (e.g., passenger car, passenger trucks, light commercial trucks, etc.) selected in the RunSpec.

2.3.3.2.4.4 Age Distribution Importer

The **Age Distribution Importer** allows the user to enter data that provides the distribution of vehicle counts by age for each calendar year (yearID) and vehicle type (sourceTypeID). This data is required for the **County Domain/Scale** RunSpec. The distribution of ageIDs (the variable for age) must sum to one for each vehicle type and year.

This tab does not have the "**Export Default Data**" button and the user is required to acquire/generate this data independently if either the **County** or **Project Domain/Scale** are chosen.

2.3.3.2.4.5 Average Speed Distribution Importer

The **Average Speed Distribution Importer** allows the user to input average speed data specific to vehicle type (sourceTypeID), road type (roadTypeID), and time of day/type of day (hourdayID). MOVES defines 16 "speed bins" which describe the average driving speed on a roadtype or link. The user must enter the fraction of driving time in each speed bin (avgSpeedFraction) for each hour/day type, vehicle type, road type, and average speed (avgspeedbinid), where the fractions sum to one for each vehicle type, road type, and hour/day type specified in the RunSpec. If desired, the user can use wildcards to apply one set of fractions to for all hour/day combinations for a specific vehicle type and road type. Note that MOVES uses a time-based distribution, whereas MOBILE6.2 used a VMT-Based speed distribution. Users may need to adjust their travel model post-processing code in order to generate the new distributions.

2.3.3.2.4.6 Road Type Distribution Importer

The **Road Type Distribution Importer** allows the user to enter data relating to the vehicle miles traveled by road type (roadTypeVMTFraction) for the domain being modeled. This data is entered as a distribution across road types (roadTypeID), where only road types selected by the user in the RunSpec are included in the distribution. The distribution must sum to one for each source type.

This tab does not have the "**Export Default Data**" button and the user is required to acquire/generate this data independently if the **County Domain/Scale** is chosen.

2.3.3.2.4.7 Ramp Fraction Importer

The **Ramp Fraction Importer** allows the user to modify the fraction of ramp driving on selected road types. Only limited access roadtypes (freeways and interstates) may have their ramp fractions modified. Thus, this **Importer** is only active if the user selects a limited access road type when defining the RunSpec.

The County Domain Manager accepts default values for this parameter, so importing ramp fractions is optional when **County Domain/Scale** is chosen.

2.3.3.2.4.8 Meteorology Data Importer

The **Meteorology Data Importer** allows the user to import temperature and humidity data for months, zones, counties, and hours that are included in the RunSpec. While the MOVES model contains 30-year average temperature and humidity data for each county, month, and hour, the user should enter data specific to the modeled location and time. Also, note that for SIP and conformity purposes, the default values are not appropriate.

2.3.3.2.4.9 Zone Road Activity Importer

The **Zone Road Activity Importer** is used only if the Custom Domain option is chosen in the **County Domain Manager**. This importer allows the user to allocate Source Hours Operating (SHO) across the zones of the Custom Domain.   In cases where there is only one zone in the county, the user should allocate activity to road type using the Road Type Distribution Importer (described above). Also, the table for the Zone Road Activity Importer should be populated with a "SHOAllocFactor" of 1.0 for each road type.

However, if the user creates multiple zones within the Custom Domain, this table allows the user to allocate the SHO across the zones. In this case, the SHOAllocFactor should sum to one for each road type. For example, if a custom domain has 2 zones, then SHOAllocFactor could be 0.5 for all road types in the first zone and 0.5 for all road types in the second, giving a total allocation of 1.0 for each road type.

This tab does not have the "**Export Default Data**" button and the user is required to acquire/generate this data independently if the **County Domain/Scale** and Custom Domain option are chosen.

2.3.3.2.4.10 Operating Mode Distribution Importer

The **Operating Mode Distribution Importer** is used only in the **Project Domain Manager.** It allows the user to import operating mode fraction data for source types, hour / day combinations, roadway links and pollutant / process combinations that are included in the RunSpec and Project domain. These data are entered as a distribution across operating modes. Operating modes are "modes"of vehicle activity that have distinct emission rates. For example, "running" activity has modes that are distinguished by their Vehicle Specific Power and instantaneous speed. "Start" activity has modes that are distinguished by the time the vehicle has been parked prior to the start ("soak time"). See EPA technical reports for additional information

on operating modes.   For a given source type, hour/day combination, roadway link and pollutant / process combination, the operating mode distribution must sum to one.

The Operating Mode Distribution Importer is required for the **Project Domain Manager** when modeling any 'non-running emission' process, and for modeling 'running emission' processes when the **Link Drive Schedules Importer** is not used. **Operating Mode Importer data** has precedent over data entered in the **Link Drive Schedules Importer** if conflicting data is entered.

2.3.3.2.4.11 Link Drive Schedules Importer

The **Link Drive Schedules Importer** is used only in the **Project Domain Manager.** It allows the user to define the precise speed and grade as a function of time (seconds) on a particular roadway link.   The time domain is entered in units of seconds, the speed variable in miles per hour and the grade variable in percent grade (i.e., vertical distance / lateral distance; 100% grade equals a 45 degree slope).

This Importer is used for the **Project Domain Manager** only when modeling 'running emission' processes when the **Link Drive Schedules Importer** is not used**.** Either an operating mode distribution or a link drive schedule must be entered for each of the user's defined roadway links.

2.3.3.2.4.12 Links Importer

The **Links Importer** is used only in the **Project Domain Manager**. It allows the user to define individual roadway links. The MOVES links need not correspond to traffic modeling "links" but each link should be uniform in its activity as described by the following parameters. Each link requires a linkID (i.e., a unique integer between 1 and 99999 that is used to reference the link in the program), countyID, zoneID, and roadTypeID (these also must be specified in the Runspec or an error will occur), the length of the roadway link in units of miles, the traffic volume on the roadway link in units of vehicles per hour, and the average speed of all of the vehicles on the roadway link in the given hour. The link description field is a text field included for reference and has no impact on the calculations.

A project may include a single off-network (parking lot or other non-road) link.   For an off-network link, the user should enter a value of zero for link length, link volume and link average speed.

2.3.3.2.4.13 Link Source Types Importer

The **Link Source Types Importer** is used only in the **Project Domain Manager.** It is used to enter the fraction of the link traffic volume which is represented by each source type. It is not used to enter off-network data, and is not required if the Project contains only an off-network link.   For each linkID, the sourceTypeHourFraction must sum to one across all source types. The user may enter data in this importer for source types which are not selected in the Runspec. However, MOVES will ignore data entered in this manager for source types which are not selected in the Runspec. The Project level calculator will not re-normalize the fractions to omit the contribution of source types which are not selected in the Runspec.

2.3.3.2.4.14 Off Network Importer

The **Off Network Importer** is used only in the **Project Domain Manager**. It provides information about vehicles are not driving on the project links, but still contribute to the project emissions. For each source type in the RunSpec, 'vehicle population' is the average number of "off network" vehicles.   . The 'start fraction' field is a number from 0 to 1.0 which specifies the fraction of this population which has a 'start' operation in the given hour.   The 'extendedIdle fraction' field is a number from 0 to 1.0 which specifies the fraction of the population which has had an 'extended idle' operation in the given hour. Finally, the 'parked vehicle fraction' field is a number from 0 to 1.0 which specifies the fraction of the 'vehicle population' which have been parked in the given hour.

For example, a vehicle population of 10 long-haul combination trucks with a start fraction of 0.5, an extended idle fraction of 0.25 and a parked vehicle fraction of 0.5 implies that the project includes the emissions associated with 5 starts, 150 minutes of extended idling and 300 minutes of parking for this sourcetype.

Note Draft MOVES2009 computes extended idle emissions only for long-haul combination trucks, not for other sourcetypes; thus the extendedIdleFraction is ignored for sourcetypeids other than 62.

Note Draft MOVES2009 does not compute evaporative emissions at the Project Domain/Scale; thus, the "parked vehicle fraction" is ignored.

2.3.3.2.4.15 I/M Program Coverage Records Editor

The I/M Program Coverage Records Editor can be accessed from either the County Data Manager or the Project Domain Manager. Use of the I/M Program Coverage Records Editor is described in Appendix A.

2.3.4 Action

Action provides a drop-down menu with the choices **Execute, Stop, Pause, Resume,** and **MOVES Run Error Log.**
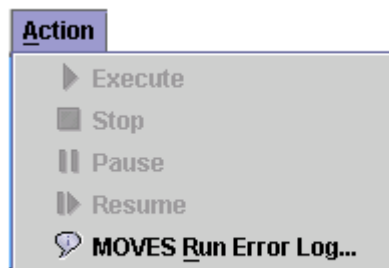
After satisfying the RunSpec input requirements (all check marks in the Navigation Panel must be set to green checks or yellow tildes - see **2.2 Navigation Panel** for an explanation of these marks), choose **Execut**e to begin the MOVES simulation. Choose **Stop** or **Pause** in the **Action** menu to stop or pause the execution of the MOVES program. These two items will only be active if the MOVES program is running. The user may also resume a paused MOVES simulation by clicking the **Resume** item.

 ⚲ Note A program that has been stopped cannot be resumed.

Click **MOVES Run Error Log** to display information on the processing outcome of the execution in the **MOVES Run Error Log** window. Alternatively, the log can be displayed by selecting the ALT and R keys when the Action menu option is open.

 ⚲ Note   This window will only appear if a MOVES Output database has been created.

Typically, the user should select **MOVES Run Error Log** immediately after the MOVES program has completed its execution. The window (shown below) will contain either a message stating that the run was successful and that no errors occurred, or it will report a brief error diagnostic. The example shows a successful run, indicating the run number (Run 5) and the time and date of the run. The user can also view the Run Error Log for the error status of previous runs by clicking the down arrow on the right side of the "**Select Run**" button.

 ⚲ Note   Not all error log messages indicate invalid results. For example, "Warning: RunSpec doesn't have all the RoadTypes", indicates a problem with the RunSpec only if the user had intended to include all roadtypes.
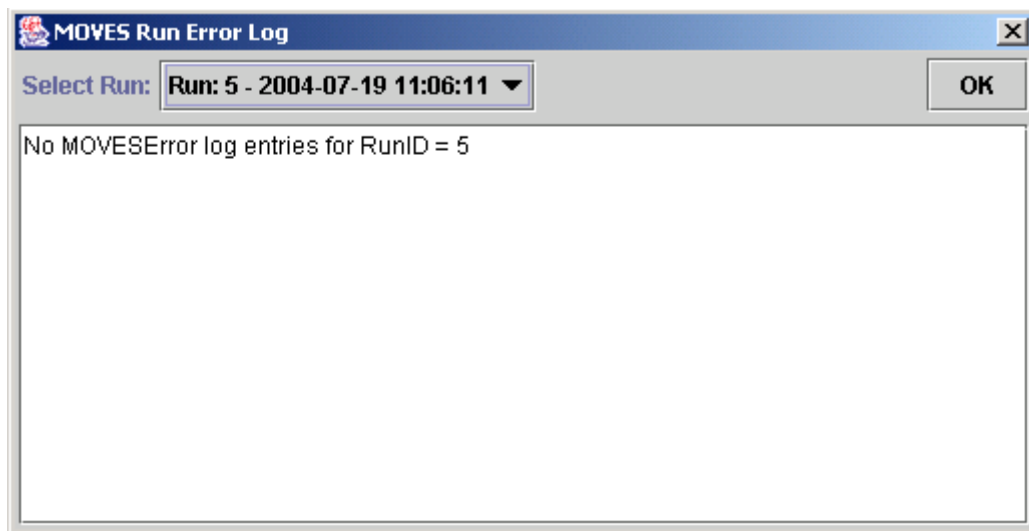
2.3.5 Post Processing

MOVES output databases can be viewed and processed directly using MYSQL Query Browser or other database tools. In addition, MOVES provides a few simple options to post-process and view the MOVES results. **Post Processing** contains three menu options, **Run**

MySQL Script on Output Database, Produce Summary Report, and **Produce State/County Map**. These features can be used after a successful execution of a MOVES RunSpec to further process the MOVES results into more aggregate or easy-to-use formats.
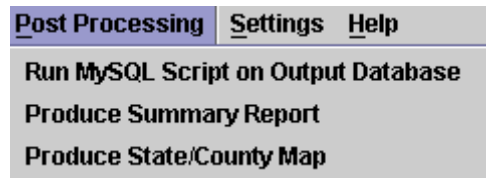
2.3.5.1 Run MySQL Script on Output Database

The **Run MySQL Script on Output Database** menu option enables the user to select from a set of MySQL scripts to further process the MOVES output databases stored in the MySQL database format. After selecting this menu option, the user clicks on the arrow in the scroll down box to view all the available scripts. After identifying the script to use, the user selects a post-processing script by clicking to highlight it. To execute the script, the user clicks the "**OK**" button.

Several scripts are distributed with Draft MOVES2009. The first script, **MOVESOutputDecodeExample.sql**" is an example script that decodes the SourceUseType and FuelType fields from numerical code classification to the full text description. The script produces two new output tables with the expanded text descriptions: decodedmovesactivityoutput and decodedmovesoutput.

The second script "**TabbedOutput.sql**" takes three MOVES MySQL output tables (MOVESOutput, MOVESActivityOutput and MOVESRun) and converts them into tab-delimited text files that can be read by a spreadsheet program such as Excel or Lotus123. This is useful if the user does not want to work with the results in the MySQL relational database format. In DRAFT MOVES2009, this can also be done with the **Summary Reporting** feature.

!Tip The **Run MySQL Script on Output Database** Post Processing feature of MOVES also allows the user to run user-created MySQL scripts. Each user-created post-processing script must be a text file containing Structured Query Language (SQL) commands, and containing a .sql file extension.   Detailed instructions on writing a MySQL script are beyond the scope of this document. To access user-created scripts from the tool bar, the completed script must be saved to the OutputProcessingScripts subfolder of the Database subfolder of the MOVES program folder. In other words, the path should be similar to the following: C:\…\MOVES\Database\OutputProcessingScripts.

2.3.5.2 Produce Summary Report

The **Produce Summary Report** menu option allows the user to summarize and report the output of one or more MOVES runs in a variety of ways. Reports can be viewed on the screen, printed, or saved in an ASCII text form that is easily imported into other software formats such as Microsoft Excel. The Summary Reporter makes it easy to display and print summary

reports of MOVES Output information and convert them to ASCII text files. For the Summary Reporter to operate properly, the run specification which produced the output, or one sufficiently similar to it, needs to be loaded in MOVES.

When **Produce Summary Report** is selected and suitable run specification has been loaded, a popup appears asking the user to specify an emission process to be reported. The default option (which is the only option if the run did not distinguish results by emission process) is to combine the output results for all emission processes included in the run specification. But in many cases the user will want to select an emission process from the drop down list. "**Evap Permeation**" is selected in this example:
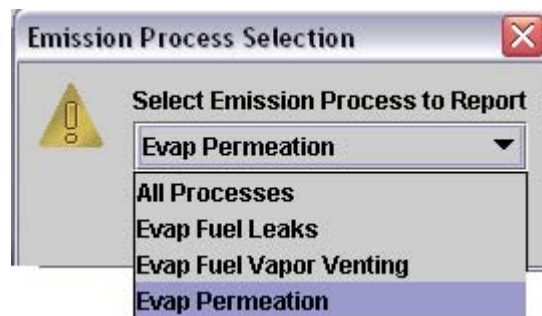
Clicking "**OK**" allows the user to specify the content and level of report aggregation.
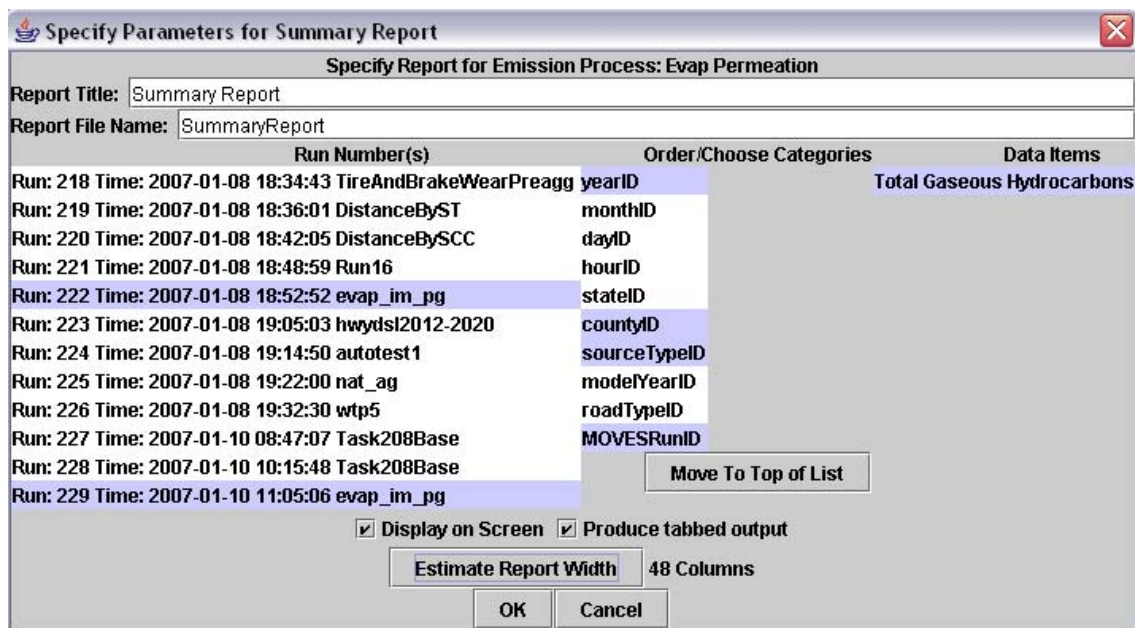
This screen allows the user to:

1. Give the report a title. The default title is "Summary Report" but the user may change this in the text field.

2. Specify the base portion of the file names which will be used to store various portions of the report. The default file name root is "SummaryReport" but the user should change this in the text field.

$\mathcal{N}$**Caution!** Files which already have this base file name will be overwritten without warning if reports are generated with the same base file name.

3. Select one or more runs to be reported. Their output must have been stored in the same output database named in the active run specification. The twelve most recent runs are available for selection. In the example shown, two runs have been selected. Note that these two runs were made with the same run specification.

$\mathcal{N}$**Caution!** Care should be taken to compare runs with the same scope and level of output detail.

4. Choose the categories to be distinguished in the report. At least one category must be selected. The category list is based on the run specification. The fewer categories selected, the more aggregate the report will be. Dimensions not distinguished when the run was made are not included. The "**Move to Top of List**" button may be used to reorder the category list. The order of the list determines the sort order and the column order used in the report. In this example, several categories, including "**MOVESRunID**" have been selected.
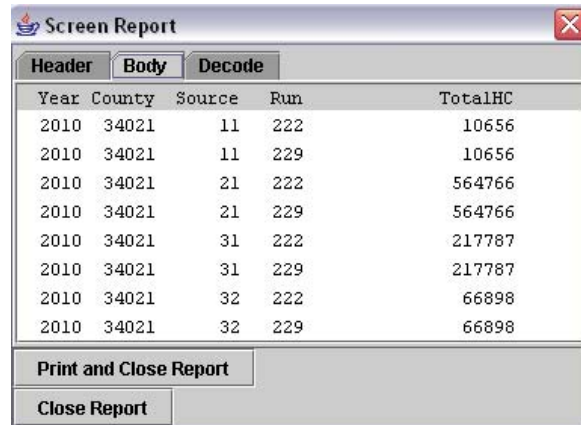
!Tip If multiple runs are included in the report, the user may wish to select the "**MOVESRunID**" category. Otherwise, the results of the multiple runs will be combined in the report. Conversely, if only a single run is included in the report, there is little reason to select the "**MOVESRunID**" category.

5. Select the data items to be included in the report. At least one data item besides "**Distance**" must be selected. In this example only one data item, "**Total Gaseous Hydrocarbons**" is available, so it must be selected.

6. Select the forms of output desired. The report always produces a set of three MySQL tables placed in the output database. There is no selection governing this. If a screen display of the report is desired, the user checks the "**Display on Screen**" box. If ASCII tab-separated variable file output is desired, the user checks the "**Produce Tabbed Output**" box.

$\mathcal{N}$**Caution!** If the output data is disaggregated by day or hour, the output database will contain results for only two days in each month - a weekday (dayID 5) and a weekend-day (dayID 2). Therefore, attempting to see a summary report for a month or year will not return valid result as these two types of days together are simply summed together. The user should distinguish by dayID in the report and then externally must properly weight the result according to the type of day to see meaningful result for a month (e.g. 22 weekdays and 9 weekend-days in a 31-day month) or year (e.g. 261 weekdays and 104 weekend-days in a 365-day year).
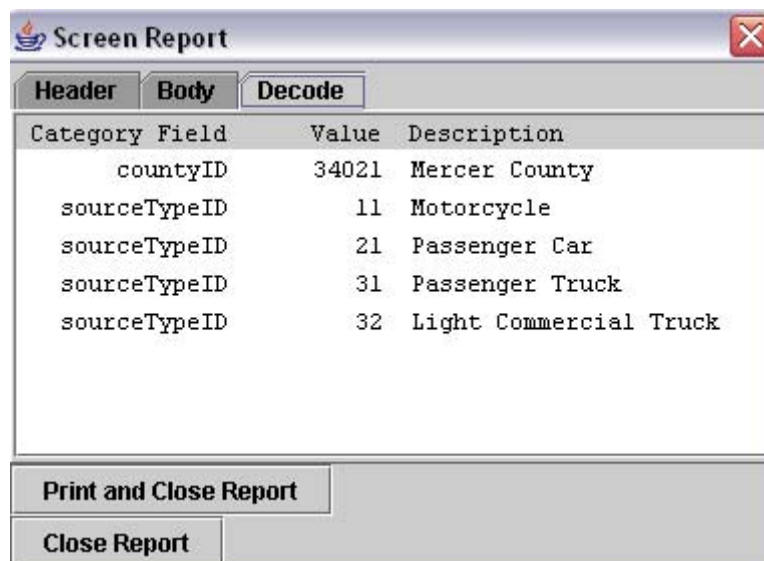
Because the width of a report can be a constraining factor (for example, when printing the report) the "**Estimate Report Width**" button can be clicked. When this is done, as in the example screen above, an estimate of the width of the report with the currently selected items appears in the window. Printed report output is obtained by producing a screen report and then choosing to print.

In the example shown above, all report options have been selected. When "**OK**" is clicked, the screen report, if requested, appears:

Note that the report consists of three tabs: **Header, Body,** and **Decode**. The **Body** tab is the default view; the other tabs are accessed by clicking on them. The body of the report contains columns for each of the category and data items selected. Clicking on the **Header** tab displays the report title, date and time the report was made, several items of information about the run specification, and the emission process covered by the report. Clicking on the **Decode** tab displays the decoded numeric codes included in the report, which are not self-explanatory.

When the user is done viewing the screen report, the user may click on "**Print and Close Report**" to print. Printing is in landscape format. Clicking on "**Close Report**" closes the report without printing it.
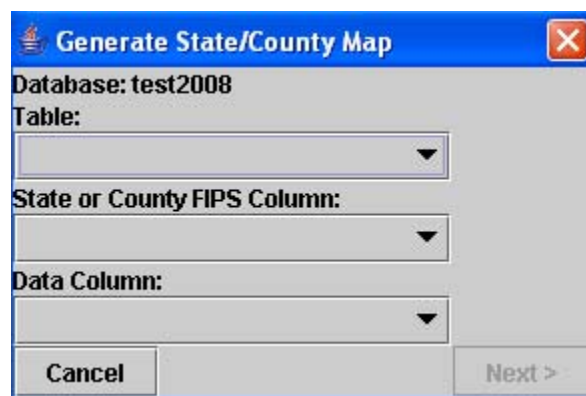
📏**Caution!** Both buttons close the report. The Summary Reporter does not store reporting options, other than the last report title and base file name used. Because information is not remembered between Summary Reporter runs, the user should not close the report until the user is done viewing it.

Once any screen report has been closed, and if ASCII tab-separated reporting has been selected, the user is prompted for a file directory in which to place the report output, and these files are written.

2.3.5.3 Produce State/County Map

The **Produce State/County Map** menu option allows the user to see the RunSpec output depicted in color-coded or gray-scaled maps. The maps can illustrate output at the county level, and represent value ranges of a single numerical variable which can be selected from a list by the user. It should be noted that this is a very basic mapping tool. For more sophisticated mapping options, you may want to apply GIS tools or other software to the MOVES database output.

For the **Produce State/County Map** tool to operate properly, the run specification which produced the output, or one sufficiently similar to it, needs to be loaded in MOVES. When the user chooses the **Produce State/County Map** menu feature, the **Generate State/County Map** window opens and displays the name of the source database for the map generation at the top of the window. (The source database for the map generation is the database containing the output of the RunSpec.)

To view RunSpec output on a map, the user must first choose a table contained in the output database that has "**CountyID**" or "**StateID**" as a column and associates each emission quantity with each county or state selected only once; in other words, if there are multiple entries for the same pollutant in the same county, then the **Produce State/County Map** tool cannot be utilized. For example, if all 24 hours were modeled and the level of output detail was chosen to

be "**Hour**", then there are at least 24 values for each pollutant in each county and the "**MovesOutput**" table cannot be used. It is possible for the "**MovesOutput**" table to be used if the user has appropriately defined or aggregated their RunSpec; however, it is more likely that the user will have to use the **Summary Reporter** to generate a table that fits the necessary criteria.
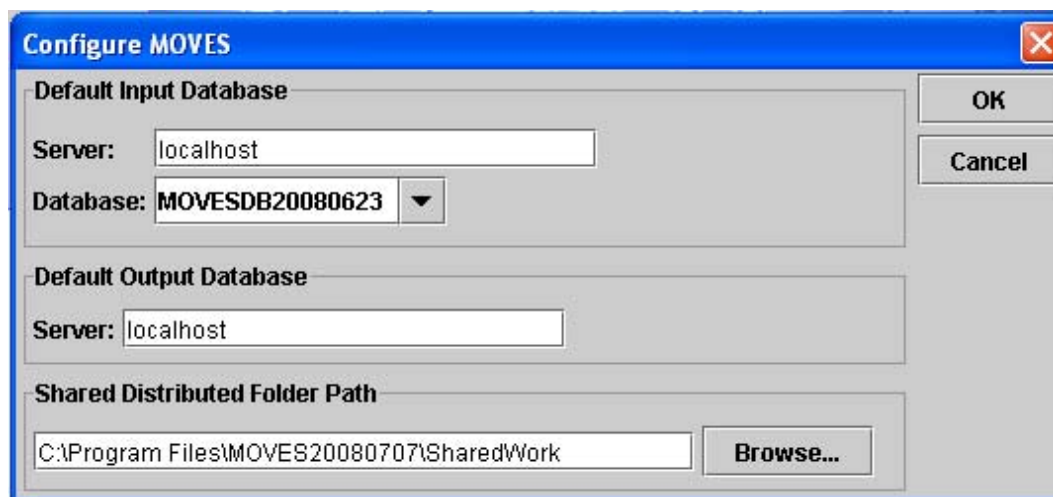
        Once a table with the attributes described above is created, it can be selected in the "**Table**" drop-down menu. The State or County FIPS Column box will be automatically populated with "**CountyID**" or "**StateID**", instructing the mapping tool to consider output data at the county or state level. Next, the user should specify data to depict by selecting an output parameter in the **Data Column** dropdown menu. By choosing a column from the selected table, the user instructs the mapping tool to use results from the RunSpec to generate the map output. After making selections in this window, the user should click "**Next**".

        A window allowing the user to define map attributes opens. The user can choose "**Full Color**" or "**Grayscale**" using the **Colors** drop-down box, and can customize the map using the check boxes and defining minimum and maximum data values for the emissions output data. When the user clicks "**Next**", the map output is generated.
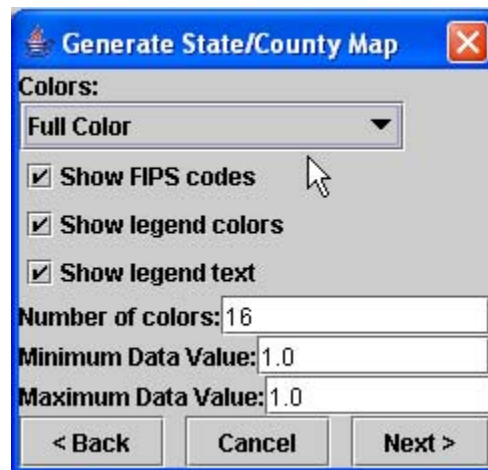
2.3.6 Settings

        Settings include one menu option, **Configure MOVES**. This menu option allows the user to set the basic MOVES operating configuration.

First, the user can specify the default input database by identifying the default server on which the MOVES input database is stored and the name of the default database. In most cases, "localhost" is the chosen server, although the user may select an alternate server (such as the MySQL database located on the user's computer) by typing the server name in the Server box. Next the user can choose from a drop-down list to define the default database to be used as a data source for the MOVES RunSpec. In most cases, the user will choose "MOVESDBDraft2009" The user also can select a server to house the default output database server under the Settings menu option; "localhost" is appropriate in most cases. The actual output database file is named when selecting Output from the Navigation Panel to specify output parameters as part of the MOVES RunSpec setup process.

Caution! If the **Default Input Database** is changed the program should be stopped and restarted. This is because the user interface program will have already connected to the previously specified database in order to have constructed the GUI displays, which might not be relevant for a different database.

Using the Settings menu option, the user also can identify the Shared Distributed Folder Path for the MOVES Worker and MOVES Master program modules. This is the Windows folder where all intermediate and internal MOVES work files are stored during processing. This folder is installed as part of the MOVES program installation. The user should browse his system to find where this folder was installed and indicate the exact location. It may not be the same path as shown in the example above.

!Tip By changing the Shared Distributed Folder path, the user can take advantage of distributed processing by having the master and multiple worker computers on the same network "point" to a drive and folder that is accessible to all the computers involved. For more information about utilizing distributed processing, please consult the Draft MOVES2009 Software Design and Reference Manual.

2.3.7 Help

Help provides access to .pdf versions of the MOVES User Guide and the MOVES Software Design and Reference Manual, and supplies general information about the version of MOVES being used.

Choose **MOVES User Guide** or **MOVES Software Design/Reference Manual** to open either of these documents in .pdf format.

Choose **About MOVES** in the **Help** menu to obtain the release date of the model. This indicates which version of MOVES is open. In addition to the version, the EPA copyright and GNU General Public License (GPL) information is provided. The GNU license grants the user, free of charge, complete access to the object and source code of the MOVES program for personal use. For specific details regarding the GNU license, the user should consult the website http:/www.gnu.org.licenses/.
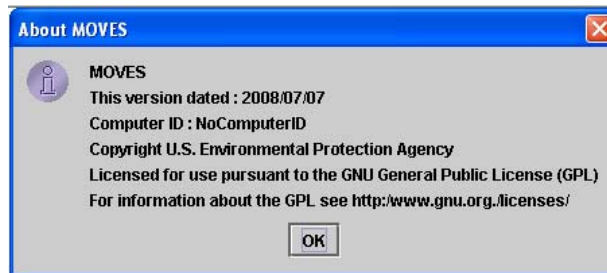
Appendix A - Displaying and Editing Vehicle Inspection and Maintenance (I/M) Program Coverage Information

The **Display/Edit IM Program Coverage Records** menu option can be accessed from the **Pre Processing** drop-down menu in the Main Menu bar or through the "**IM**" tab in the **County Data Manager** and **Project Domain Manager**. This menu option gives the user access to a set of features called the **I/M Coverage Editor**, which allows the user to display and modify vehicle inspection maintenance (I/M) program coverage information.
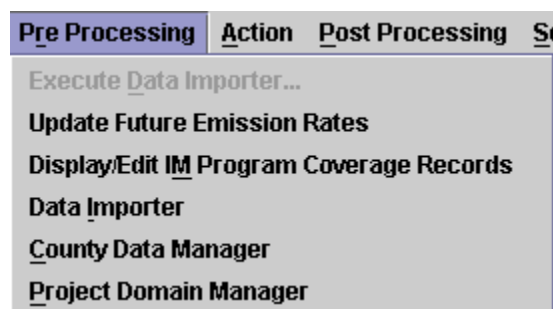
Although current local data may be more accurate, the MOVES default database is intended to contain I/M coverage data on a county/year/pollutant/fuel type/vehicle type basis for all I/M programs in the United States. The defaults will automatically be used in MOVES runs unless the I/M Coverage Editor is used to create new user input database that overlays the default IM coverage data.   If the user opens the I/M Coverage Editor directly, the new user input database will be listed in the MOVES **Manage Input Datasets** panel. If the user opens the I/M Coverage Editor through the County Data Manager or the Project Domain Manager, the modified I/M coverage data will be stored in the database created by that tool, as entered on the **Geographic Bounds** panel.

The **I/M Coverage Editor** requires that the MOVES RunSpec specify the calendar years, locations (states or counties), source use types, and pollutant-processes of interest. The **I/M Coverage Preprocessor** will not open until these MOVES RunSpec inputs have been made.

!Tip   The user is encouraged to successfully complete (i.e., green checkmarks) all of the main MOVES RunSpec check-list items before accessing the **I/M Coverage Editor**.

When the **Display/Edit IM Program Coverage Records** menu option is selected, MOVES will show the **IM Coverage in MOVES** window.

This panel displays the I/M coverage information pertinent to the RunSpec. A scroll bar enables the user to see the complete list of the I/M coverage information, which includes the following fields:

- **Process.**

⚲ Note   MOVES I/M emission effects are applied to Running Exhaust and Evaporative Fuel Vapor Venting processes. I/M Emission effects for Starts are not functional in Draft MOVES2009.

- **Pollutant.**

⚲ Note   MOVES I/M emission effects are applied only to HC, CO and NO_X pollutants.

- **County.**

- **Year-- that is, the calendar year in which the IM program applies**

- **Fuel-- that is, the fuel-type of the vehicle subject to the program.**

- **Regulatory class.**

- **Old Vehicle Age/New Vehicle Age** - These fields specify the age range covered in the chosen I/M program. The allowable range is zero to thirty years (years beyond thirty can be entered and performance will not be affected).

- **Inspection Frequency** - The three possible inputs for **Inspection Frequency** are annual, biennial, and continuous.

✏ CAUTION!   I/M emission effects are not currently available for a continuous I/M program.

- **Compliance Factor** - The **Compliance Factor** reflects the level of motorist compliance that the particular I/M program achieves. Different levels of compliance can be entered for each I/M program identification number. Only values between 0 and 100 are valid.

⚲ **Note**   The I/M **Compliance Factor** is a multiplicative factor that adjusts a particular I/M program for losses associated with motorists who do not comply with the requirements of the I/M program. Typically, this includes motorists whose vehicles fail I/M but are not repaired or insufficiently repaired and continue to operate in the program area, as well as motorists whose vehicles are subject to I/M but are never tested.

- **Test Type/Test Standards** - The **Test Type** and **Test Standards** selected should be consistent with each other (i.e., ASM (Acceleration Simulation Mode) test type with ASM cutpoints, transient test type with IM240 cutpoints, evaporative test type with evaporative tests). Failure to select consistent test types and test standards will result in error messages later.

- **Pgm ID** - The **Pgm ID** column contains a program identification number to distinguish multiple I/M programs applying to the same vehicles. An I/M program is a combination of beginning and end model years, inspection frequency, test type, test standards and compliance factor. The Pgm ID must be an integer from 1 to 999. If multiple I/M programs apply to the same combination of county, state, year, pollutant, process, fuel, and regulatory class, each program must have a distinct program ID.

- **Ignore** - The **Ignore** column is a Yes/No toggle for the use of the I/M coverage information, and can be modified by using the "**Ignore I/M**" and "**Use I/M**" buttons described later in this section. A value of 'N' is the default and signifies that MOVES will use the I/M coverage information. A value of 'Y' means either that there is no I/M program or that it is completely ineffective (i.e., non-I/M emission rates are used in the MOVES emission calculations). In this case, MOVES will ignore the chosen record when calculating I/M coverage effects, effectively creating a no I/M condition for the chosen combination of county, year, fuel type, regulatory class, and pollutant-process.

!Tip Setting all of the I/M Coverage records to Ignore = 'Y' will completely shut off the effect of I/M for the RunSpec.

The lower panel in the **IM Coverage in MOVES** window includes a message panel that displays error messages if the user input data is incorrect. The middle panel contains the following buttons:

Print Report
Edit Selected Records
Generate New and Replacement Records
Export
Import
Ignore I/M
Use I/M
Delete
Exit

These are described below.

Print Report Button: The user may print a report of the contents of the **IM Coverage in MOVES** window by clicking the "**Print Report**" button, which brings up a dialog box with standard Microsoft Print functions. The user may choose to print the information to a printer or other virtual network device.

Edit Selected Records Button: The "**Edit Selected Records**" button will be inactive until the user selects a record or set of records from the **IM Coverage in MOVES** window. The user may select a single record by clicking on it.

✎**Caution!** The "**Edit Selected Records**" button should be used to edit only one record at a time. To edit multiple records at once, the user can use the "**Generate New and Replacement Records**" button as described later in this section.

When the user selects one or more records, the "**Edit Selected Records**" button becomes active and may be used to change the data values in the selected records. If this button is selected, a window appears asking the user to **Choose a User Input Database**.
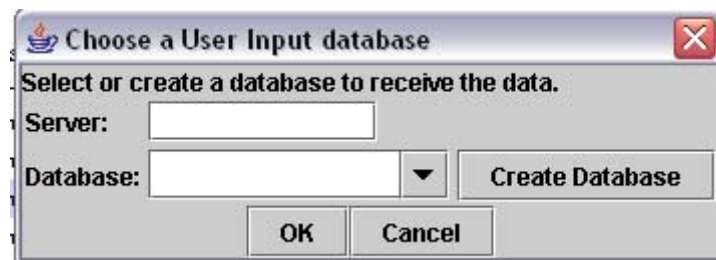
📍**Note** If the **I/M Coverage Editor** is accessed through the **County Data Manager** or **Project Domain Manager**, the associated user input database is used.

The user should normally leave the server name field set to "localhost". An existing user input database, which must contain an **I/M Coverage Table**, can be specified in the database field, or a new database name can be entered and created by clicking on the "**Create Database**" button. A window will appear confirming that the database was created.

📍**Note** Operations in the **I/M Coverage Editor** create an overlay database extending or superseding the values in the default database. These operations do not change the default database values.

✎**Caution!** The user should be careful when selecting existing databases from the list of databases in the "Choose a User Input database" window. The **I/M Coverage Editor** tools have the ability to alter data without an 'undo' function in any MySQL database that contains an I/M Coverage table (except the current, active MOVES input database). An input or delete error may lead the user to inadvertently modify previous work.

Clicking the "**Cancel**" button in the **Choose a User Input Database** window aborts the editing operation. Clicking on the "**OK**" button will cause the program to display the **IM Coverage Details** window.

The **IM Coverage Details** window allows the user to enter program data that will substitute for the default values in a particular I/M Coverage record. These fields are the same at those in the **IM Coverage Details** window, described above.

Once the user has made any edits, clicking on the "**OK**" button enters the new values into the user's I/M Coverage database, while clicking on the "**Cancel**" button closes the **IM Coverage Details** panel without saving the new data.

Generate New and Replacement Records: The **Generate New and Replacement Records** is similar to the **Edit Selected Records** in its effect, but it allows the user to change multiple records at the same time.

When the "**Generate New and Replacement Records**" button is selected, the **Choose a User Input Database** window shown earlier appears.

 Note   If the **I/M Coverage Editor** is accessed through the **County Data Manager** or **Project Domain Manager**, the associated user input database is used.

The user should normally leave the server name field set to "localhost". An existing database, which must contain an I/M Coverage Table, can be specified in the database field, or a new database name can be entered and created by clicking on the "**Create Database**" button. A window will appear confirming that the database was created.

Clicking on the "**Cancel**" button in the **Choose a User Input Database** window aborts the editing operation. Clicking on the "**OK**" button causes the program to display the **IM Coverage Categories** window, which allows the user to enter larger amounts of alternative I/M Coverage data at one time into a MOVES user database.

!Tip The user may have to expand the **IM Coverage Categories** window to see the text in each
column.

This window contains five columns of I/M coverage data - **Pollutants, Counties, Years,
Fuels,** and **Reg(ulatory) Class** - displaying only those entries selected by the user in the general
MOVES RunSpec. Alternate I/M coverage records cannot be generated for pollutants, counties,
years, fuels and regulatory classes that have not been already specified by the user in a MOVES
RunSpec. Within each column, the user may either choose all of the entries using the "**All**"
buttons or select individual entries using a mouse. The user must select at least one entry in each
column. The user must also specify a value between 1 and 999 in the "IM Program ID" box, and
should choose "IM Program ID" values that have not been used already if a new I/M program is
being created (see the **IM Coverage in MOVES** window).

 Note   The user can select an existing "IM Program ID" to modify the I/M program with that
ID.

Clicking on the "**OK**" button opens the **IM Coverage Details** panel described above.
Clicking on the "**Cancel**" button closes the **IM Coverage Categories** window without saving the
new data.

!Tip MOVES does not allow more than one I/M coverage record for each combination of
pollutant, process, county, year, fuel type, regulatory class, and model year. The duplicate
record error, "Error: Duplicate Active Program", is common, and often indicates an issue
with multiple programs applying to the same model year. Careful review of the oldest and
newest vehicle ages in your input file may help eliminate this error.

Export Button: The "**Export**" button allows the user to export the selected I/M
coverage records to an external file or spreadsheet. An error message will result if no records are
selected and the "**Export**" button is selected. If the "**Export**" button is selected and I/M
coverage records are selected, the **Export IM Programs** window will open. The user may select

the folder and file in which to store the I/M coverage data.   The data will be saved in tab-delimited format or the user can define the file path to save it as an Excel or text file. The primary function of the "**Export**" button is to allow the user to export selected records to an external file (e.g., a Microsoft Excel spreadsheet) for editing. The "**Export**" button will automatically export the correct names and spelling for the process, pollutant and county names. After editing the I/M coverage records, the user will typically import the data back into MOVES using the I/M "**Import**" button.

   Import Button: The I/M "**Import**" button allows the user to import I/M coverage records from an external text file or spreadsheet into a user input database. Selecting the "**Import**" button opens the **Import IM** window, allowing the user to select the folder and file where the desired I/M coverage data are stored. These files may be tab- or comma-delimited text, or in Microsoft Excel XLS files. If an Excel workbook with multiple worksheets is selected, MOVES will prompt the user to select the desired individual worksheet within the workbook.

   Next, the **Choose a User Input database** window will prompt the user to select the database where the imported I/M data will be written. The server name field should normally be "localhost". An existing database with an I/M Coverage Table can be specified in the database field, or a new database can be created by entering a new database name and clicking the "**Create Database**" button. A panel will confirm that the database was created.

   The I/M data importer requires that the import data file have a full set of correct column headings. For most of the inputs, the user must use the abbreviated name (aka "shortName). Gasoline vehicles should be indicated with the text "Gasol". The I/M data importer will also generate an error message - "Error: Duplicate Active Program" - if there are duplicate data errors. As with the "**Edit Selected Records**" and the "**Generate New and Replacement Records**" buttons, the user can enter no more than one I/M coverage record for each combination of pollutant, process, county, year, fuel type, regulatory class, and model year.

!Tip To generate an I/M data file for input, we recommend first exporting the desired records. The export utility will automatically export the correct column headers and default data values. Users can then use a spreadsheet or text editor to make data changes, such as changing compliance rates. The user can refer to the "emissionprocess", "pollutant", and "regulatoryclass" tables in the MOVES default database, and the "imtesttype", and "imteststandards" tables below to see the available options for the "Process", "Pollutant", "Regulatory Class", "Test Type", and "Test Standards" columns, respectively. The appropriate "shortName" for each column should be entered, except for "Regulatory Class" where the "regClassName" should be used. Once changes are made, the data file can be imported with the "**Import**"button.

MOVES I/M Test Types

testTypeID	testTypeDesc	shortName
1	Exhaust Basic Test (Idle, 2500/Idle, Loaded/Idle)	Idle Basic
2	Exhaust Acceleration Simulation Mode Test (ASM)	ASM
3	Exhaust Transient Test (IM240, IM147)	IM240/IM147
4	Evaporative Emissions Tests (EVAP)	Evaporative
5	Exhaust Onboard Diagnostics Test (OBD)	OBD

MOVES I/M Test Standards

testStandardsID	testStandardsDesc	shortName
11	Unloaded Idle Test	Unloaded Idle
12	Two-mode, 2500 RPM/Idle Test	2500 RPM/Idle
13	Loaded / Idle Test	Loaded/Idle
21	ASM 2525 Phase-in Cutpoints	A2525 Phase
22	ASM 5015 Phase-in Cutpoints	A5015 Phase
23	ASM 2525/5015 Phase-in Cutpoints	A2525/5015 Phase
24	ASM 2525 Final Cutpoints	A2525 Final
25	ASM 5015 Final Cutpoints	A5015 Final
26	ASM 2525/5015 Final Cutpoints	A2525/5015 Final
31	IM240 Phase-in Cutpoints	IM240 Phase
32	IM147 Phase-in Cutpoints	IM147 Phase
33	IM240 Final Cutpoints	IM240 Final
34	IM147 Final Cutpoints	IM147 Final
41	Evaporative Gas Cap Check	Evp Cap
42	Evaporative System Pressure Check	Evp Pressure
43	Evaporative System OBD Check	Evp OBD
44	Evaporative Gas Cap and Pressure Check	Evp Cap,Prs
45	Evaporative Gas Cap and OBD Check	Evp Cap,OBD
46	Evaporative Pressure and OBD Check	Evp Prs,OBD
47	Evaporative Gas Cap, Pressure and OBD Check	Evp Cap,OBD,Prs
51	Exhaust OBD Check	Exhaust OBD

Delete Button: The "**Delete**" button allows the user to delete previously modified or imported I/M coverage data from a user input database, thus reversing the effects of a data import using the "**Import**" button. It does not allow the user to delete I/M coverage records from the default MOVES database. At least one record must be selected from the **IM Coverage in MOVES** window before being deleted using the "**Delete**" button.

After of the user selects one or more records and clicks on the "**Delete**" button, the **Choose a User Input database** window shown earlier will open.

♀ Note If the **I/M Coverage Editor** is accessed through the **County Data Manager** or **Project Domain Manager**, the associated user input database is used.

The user should normally set the server name field to "localhost." The existing user input database in which the selected record(s) reside will be visible in the "Database" field. Selecting the "**OK**" button will delete the selected records, while selecting the "**Cancel**" button will cancel the delete process and close the **Choose a User Input database** window.

♀ Note After a record is deleted using the "**Delete**" button, it is removed from the user input database but not the default MOVES database. As a result, the user will often notice that the I/M coverage record displayed in the **IM Coverage in MOVES** panel has been changed from a user-supplied value to the default value. The number of records appearing on the **IM Coverage in MOVES** screen will often stay the same.

Ignore I/M Button: To "turn off" I/M for one or more records in the **IM Coverage in MOVES** window, the user can highlight record(s) using the mouse and the Shift key (for more

than one record), and then click the "**Ignore I/M**" button. The **Choose a User Input database** window will allow the user to select the server and database where this change will be written. The server should generally be left at 'localhost', and the user can select an existing database which contains an IMCoverage Table, or create a new one by entering a name and clicking the "**Create Database**" button. A panel appears confirming that the database was created. Clicking the "**OK**" button closes this panel. This action will not modify the default MOVES input database. If the user sets all of the I/M Coverage records to Ignore = "Y", the effect of I/M will be shut off for the RunSpec.

Use I/M Button: As with the "**Ignore I/M**" button, if the user wants to change the status of records in the **IM Coverage in MOVES** window from "Y" to "N", the user can highlight record(s) using the mouse and the Shift key (for more than one record), and then click the "**Use I/M**" button. The user will have to select a server and database to which to write the changes, as is described in the instructions for the "**Ignore I/M**" button.

Exit Button: The user can click on the "**Exit**" button to close the **IM Coverage in MOVES** window.

Use of the I/M Compliance Rate in MOVES: The I/M compliance rate is a discounting factor expressed as a percentage that accounts for less than perfect operation of a real-world I/M program. It attempts to account for emission benefit losses associated with motorists who refuse to comply with the I/M program by not getting their vehicle tested, losses from failing vehicles that are either not repaired or inadequately repaired, and other miscellaneous losses. The MOVES model contains default values for I/M compliance rate for most I/M programs in the United States. These default values were obtained from the 2005 EPA National Emission Inventory data files. The distribution of I/M compliance rates range from 83 percent to 100 percent, with more than half of the I/M programs having a value of 93 percent - a 7% loss of potential benefit. The **I/M Coverage Editor** in MOVES allows the user to modify the default values in the range from zero percent (if the I/M program is completely ineffective) to 100 percent (signifying no losses).

♀ Note EPA plans to provide guidance on developing appropriate local I/M compliance rates.

I/M Compliance and Geographic Pre-Aggregation in MOVES: If the user selects **National Domain/Scale** in MOVES, the I/M data can be pre-aggregated to the state, or national level. If the user selects "**State**" or "**Nation**" inputs on the **Geographic Bounds** panel, MOVES will calculate emissions as an appropriate weighted average of the applicable I/M and non-I/M emission rates.

[This page intentionally left blank.]

Appendix B - Linking MySQL Tables from MS ACCESS or MS Excel (includes ODBC usage)

This Appendix describes in detail how the user can transform MOVES MySQL tables into Microsoft ACCESS database tables or Microsoft Excel spreadsheets. This may be useful for those who wish to view or manipulate these tables in an alternate format. This appendix also explains how to use the Open Database Connectivity interface protocols and tools.

Adding a MySQL database as an ODBC User Data Source

1. Install MySQL driver, MyODBC-3.51.06.exe.

2. Create an ODBC data source with the same name as the folder which corresponds to the MySQL database.

   a. In the Control Panel, select "Administrative Tools":

b. In the "Administrative Tools" menu, select "Data Sources (ODBC)":
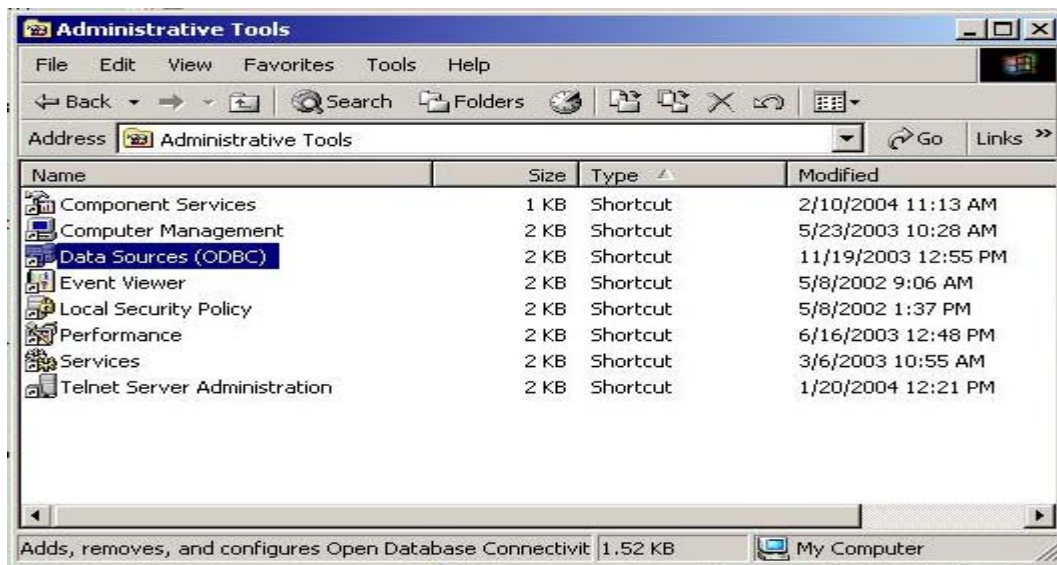
c. In the ODBC Data Source Administrator dialog choose to "Add" a "User Data Source."
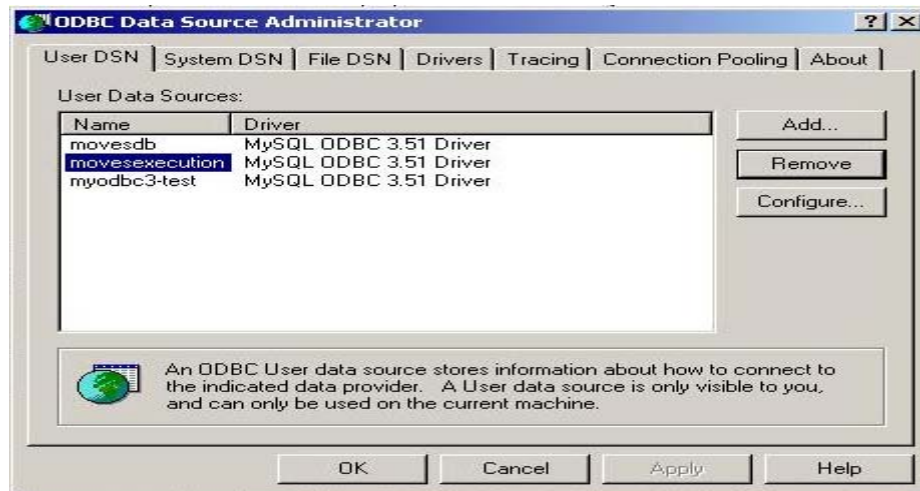
d. The "New Data Source" should use the MySQL ODBC Driver that was installed earlier (step 2a.). Scroll through the list until the MySQL ODBC driver is found, highlight it, and click the "Finish" button.

e. Next, the database links need to be defined. The values shown below are for a database called "movesdefault". The database name should be the name of the folder which contains the desired MySQL database files (i.e., the MySQL database name).
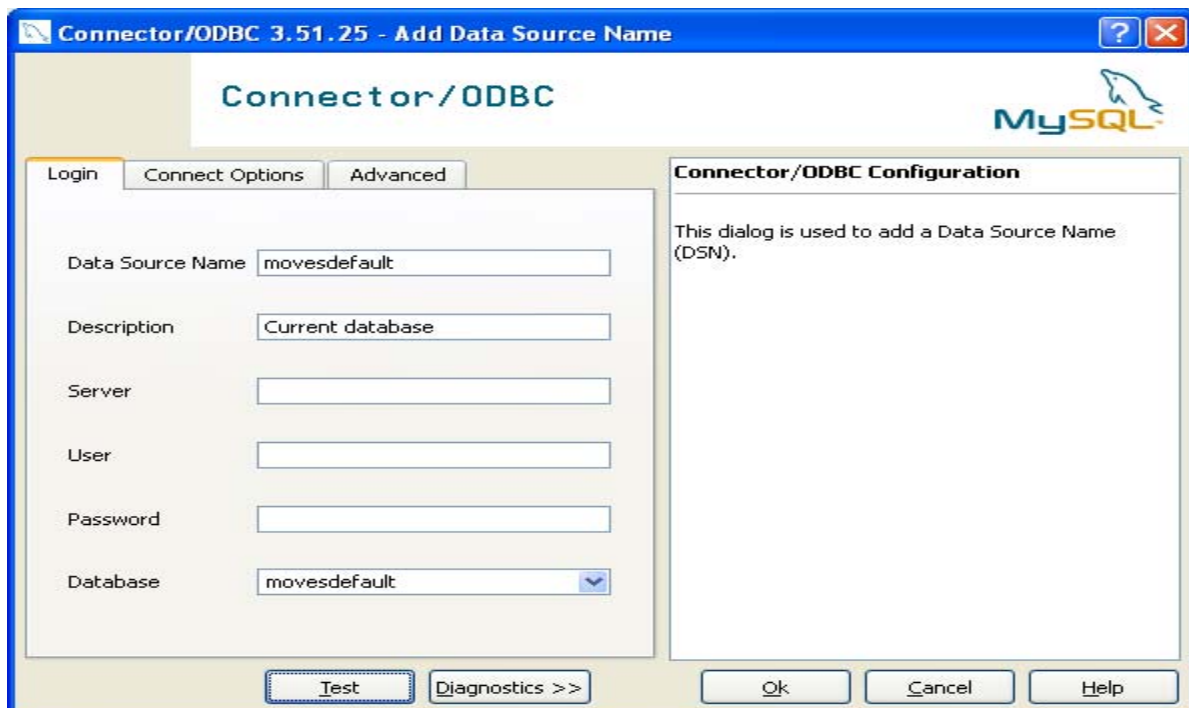
f. Once the database name and the data source name have been changed to an existing MySQL database name, the database connection can be checked by clicking on the "*Test Data Source*" button.

g. The added data source will show up in the list of data sources in the ODBC Data Source Administrator dialog. This will finish the process of creating a new ODBC data source associated with a particular MySQL database.
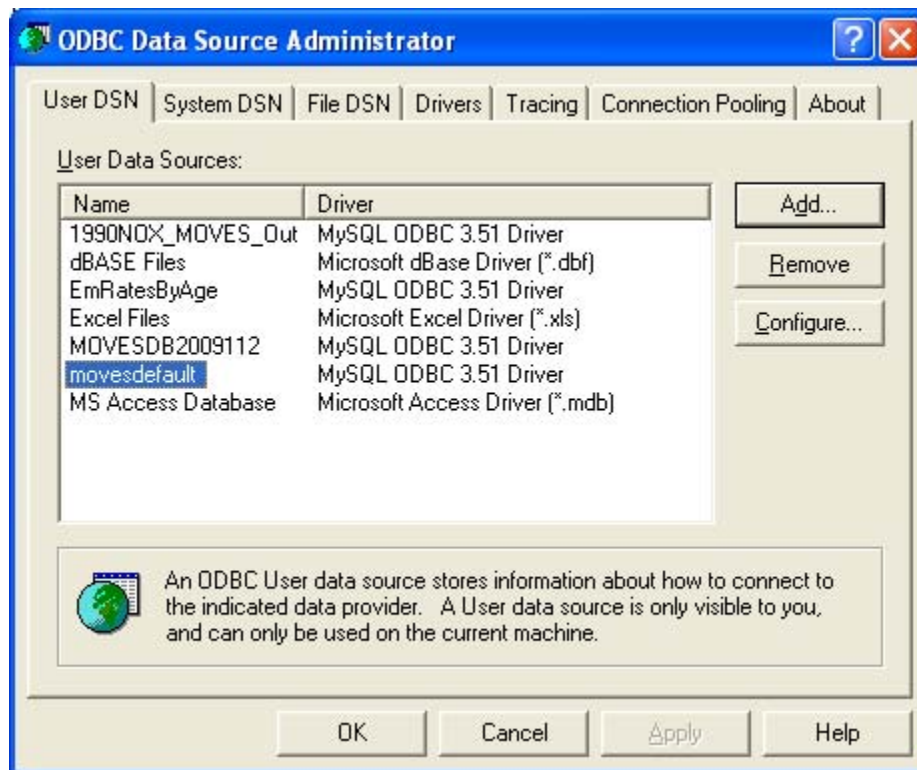
Using Microsoft Access with MySQL Tables

1. Run an Access .mdb (either a new one or an existing one).

2. On the top menu bar, look for menu items "**File**", "**Get External Data**", then "**Link Tables …**" (see picture below). Click on the item "**Link Tables…**"
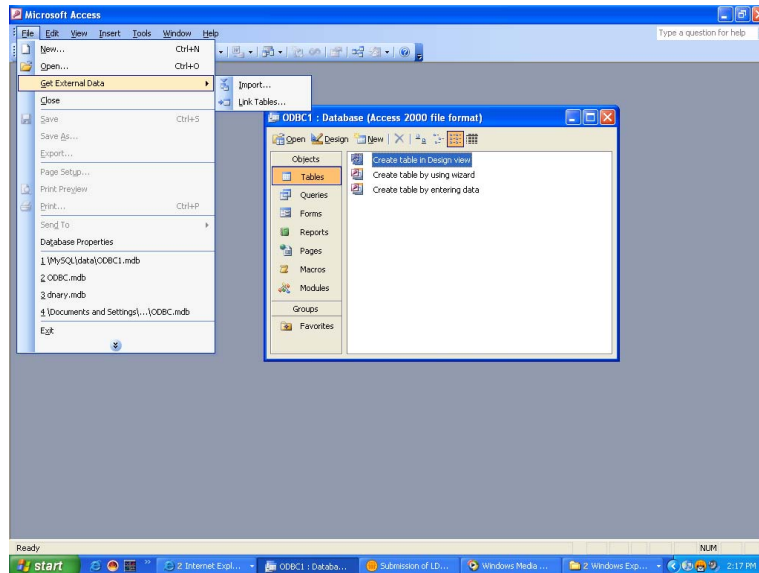
3. On the pop-up window like the one shown below, choose "**ODBC Databases**()" from the drop-down list which is located next to the label "**Files of type**".
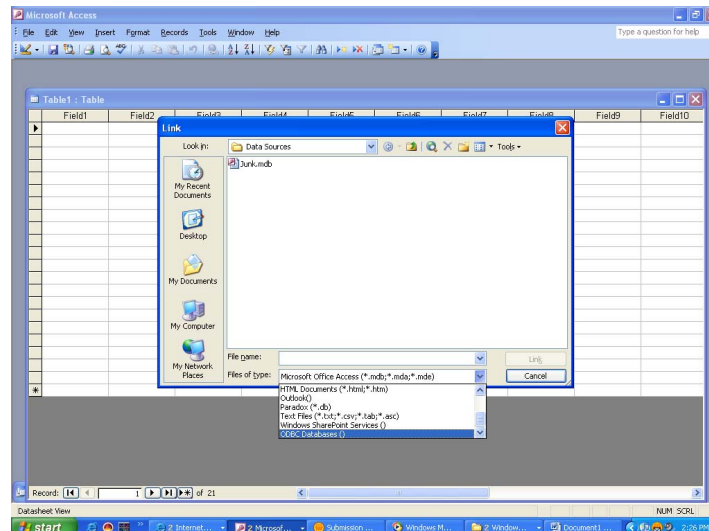
4. Another pop-up window similar to the one below will appear. Click on the tab **Machine Data Source**, to select a database of interest.
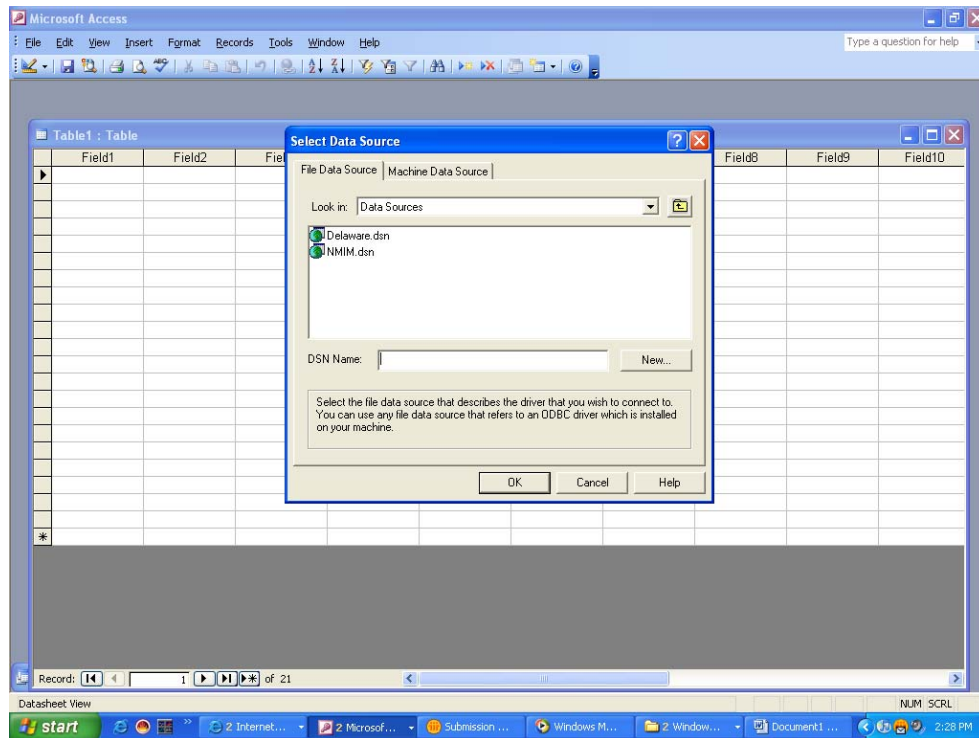
In this example NMIM20070812 is chosen from the **Data Source Name** column. Select **OK** to load the data into Access.
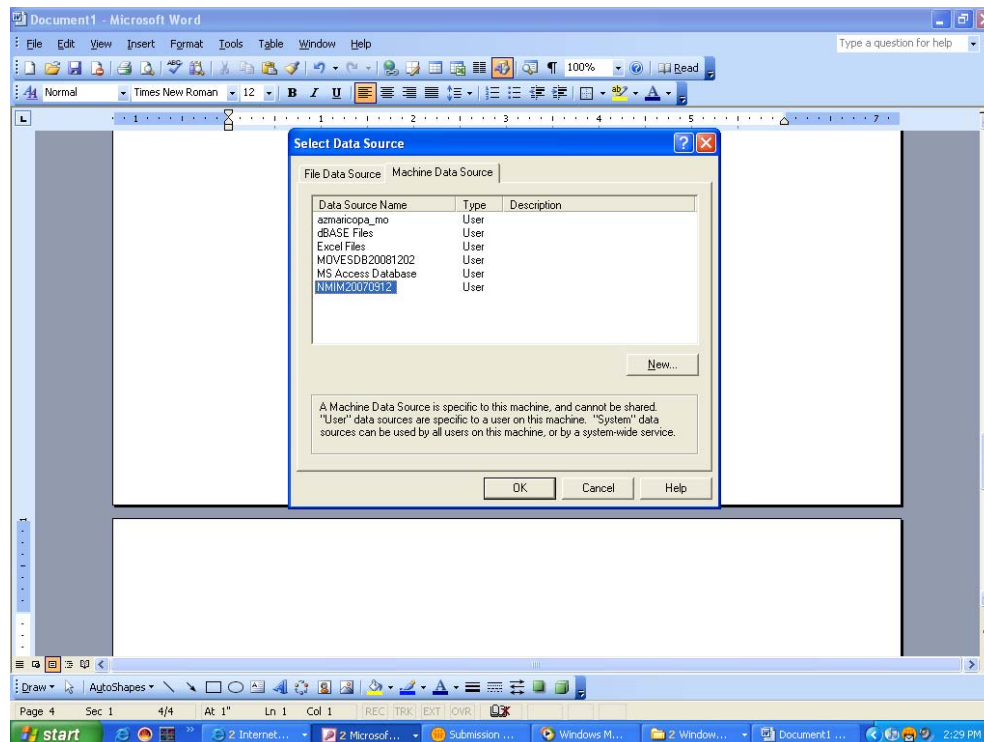
If the **Data Source Name** of the file is not listed, the user must follow the instructions provided in the section, "How to create a DSN Connection to a MySQL database" presented later in this appendix.

5. Once the data is loaded into access, the "**Link Tables**" window shown below will appear. This screen displays all of the database tables that are visible in Access. The user may select tables to view, and click the "**OK**" button.
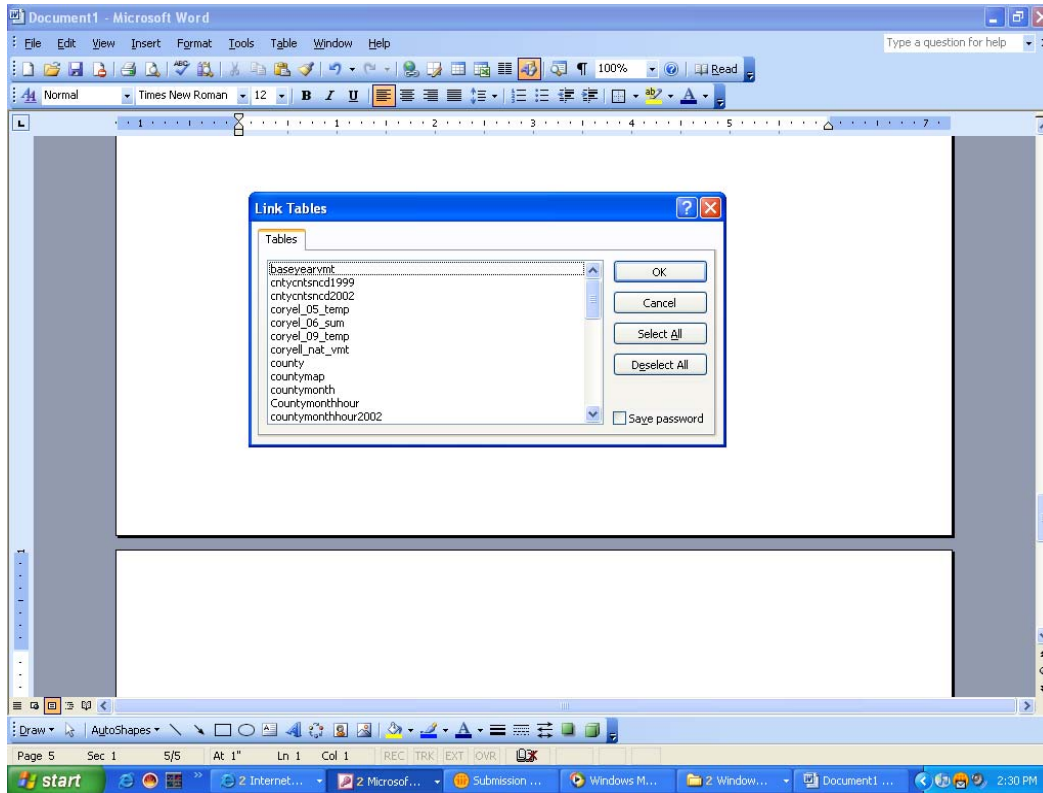
In the screenshot shown below, the user has chosen three tables to work with in MS Access: baseyearvmt, county, and state. The user can now work with these tables in MS Access.

📏 **Caution!** Edits made to the tables or data in MS Access will not affect the original MySQL tables and can not be used directly in MOVES.
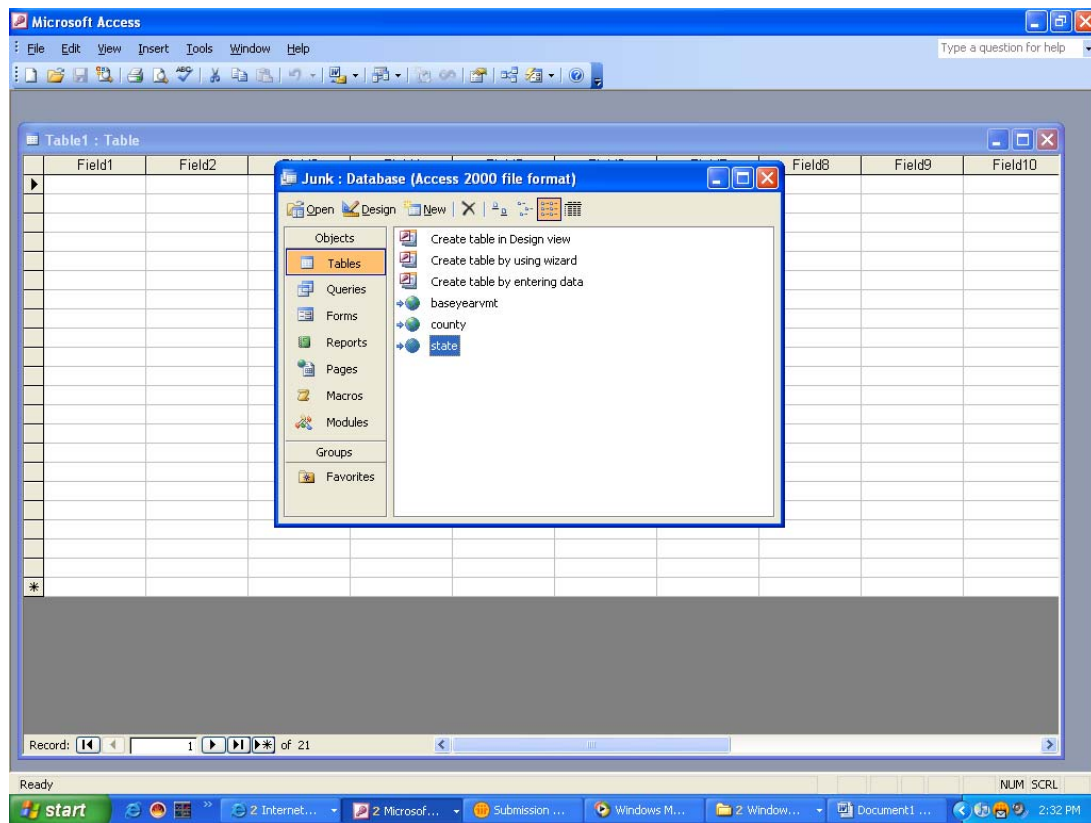
The user can view data specific to a particular state by clicking on the StateName in the table.
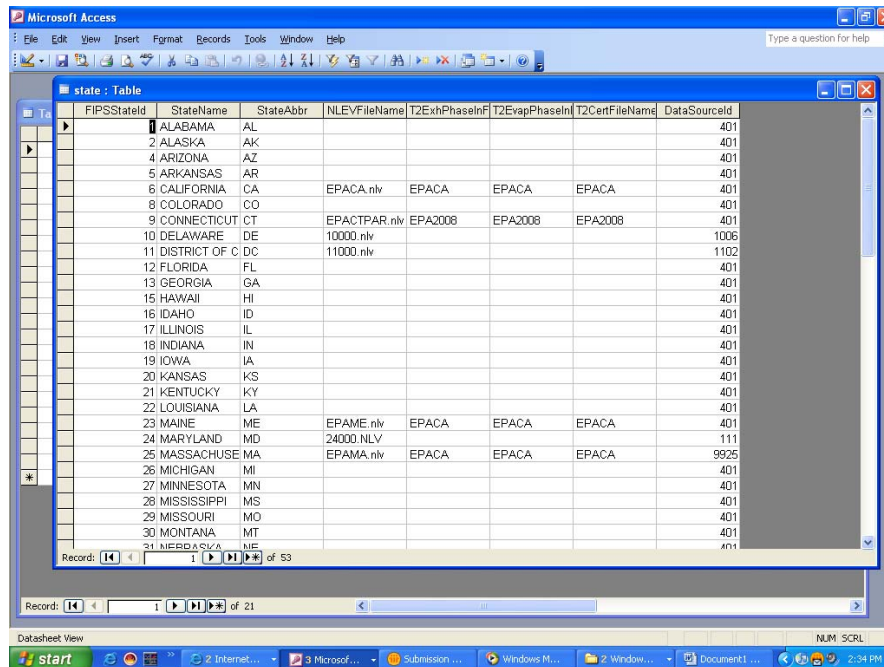
How to create a DSN Connection to a MySQL database.

If the user does not find the MySql database referenced in the **Data Source Name** listing in MS Access, a DSN connection must be established. In order to do so, click on the **Machine Data Source** tab from the **Select Data Source** screen. Click the "**New**" button, and select "**Next>**".
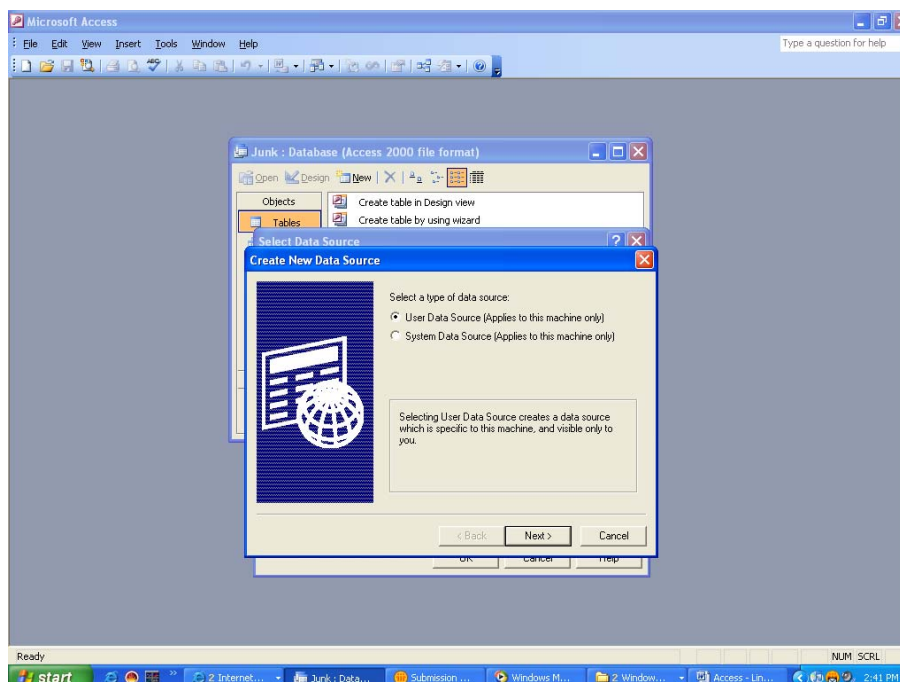
At the Create New Data Source Dialog box, select MySQL ODBC 3.51 Driver. Click "**Next >**".
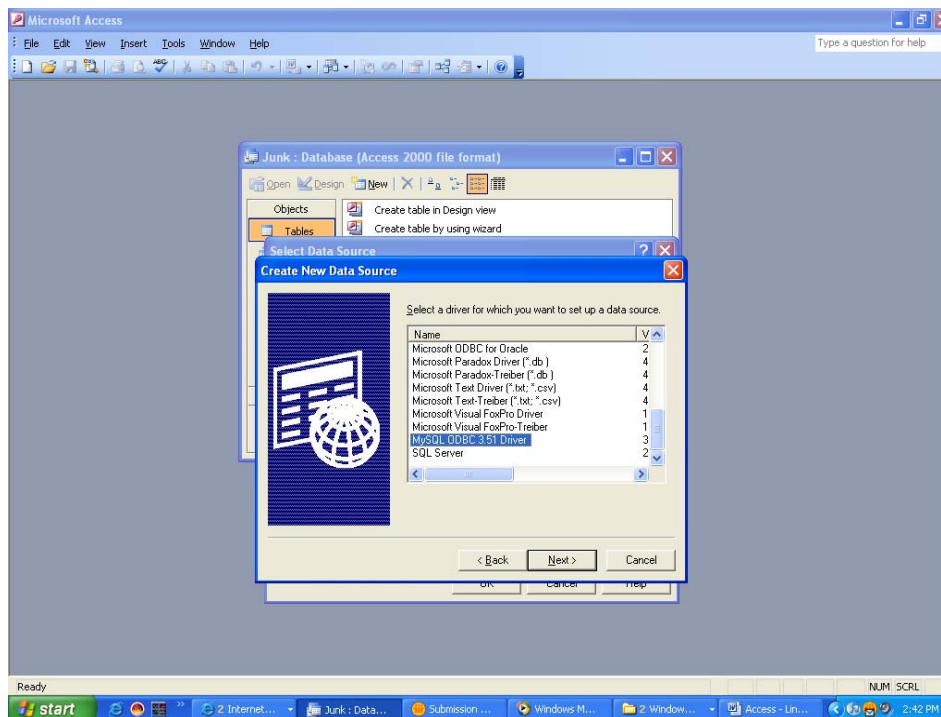
Click "**Finish**" to begin to establish the connection.
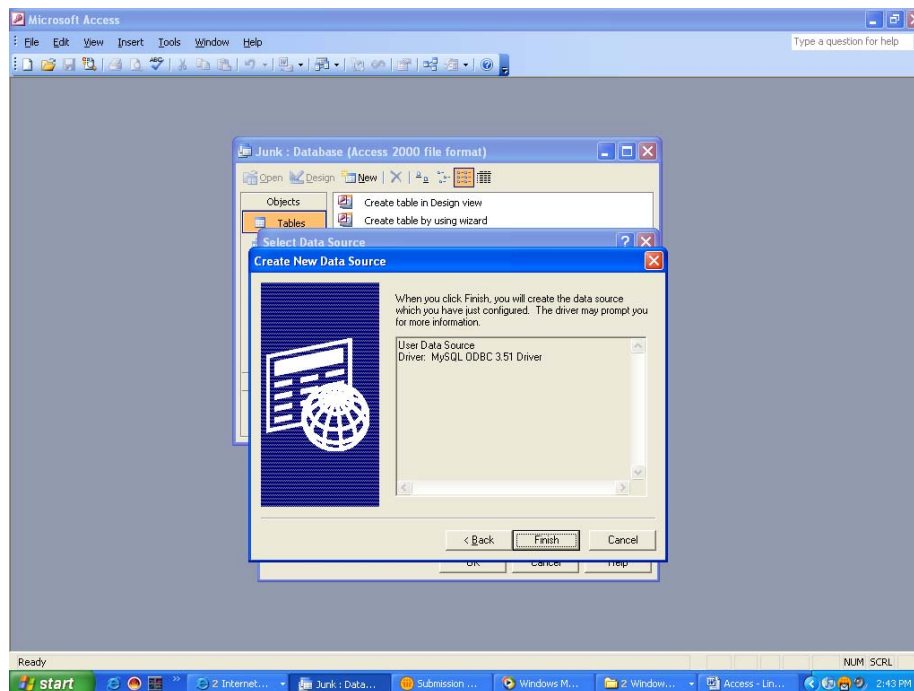
The **Connector/ODBC** screen is used to establish the connection from MS Access to the MySQL database. Use this screen to define the "**Data Source Name**".

♀ **Note**   Consider making the **Data Source Name** similar to the database name, such as DSN_DatabaseName. This will make it easier to find.
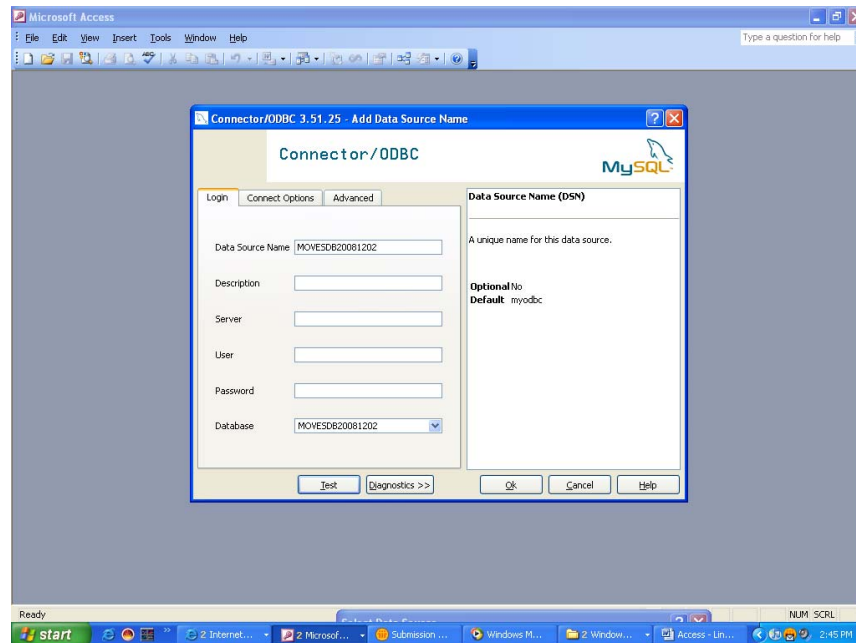
Once the **Data Source Name** has been created, select the "**Test**" button. The popup box **Connector/ODBC** should open, with the message, "Success; connection was made!" Press **OK**, and the popup box will disappear. Click **OK** to create the DSN connection.

.
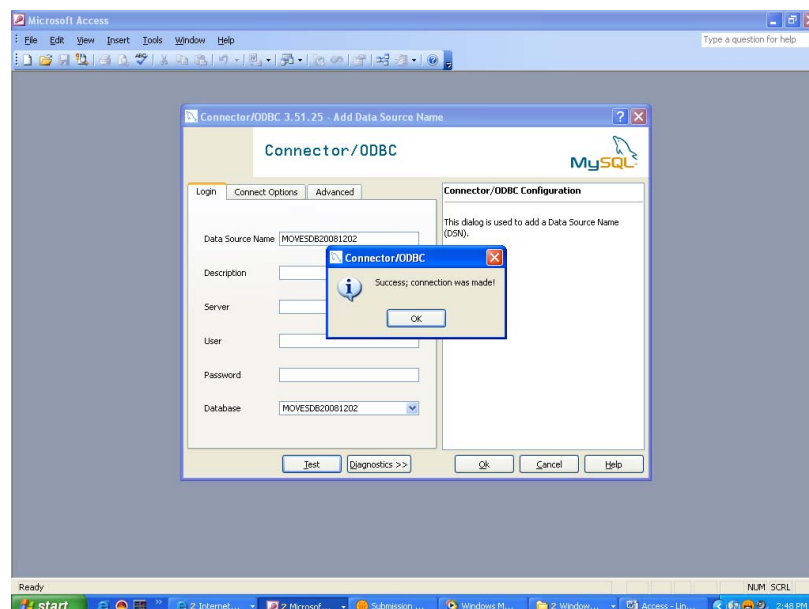
The **Data Source Name** will now be listed in the **Select Data Source** window.

Using Microsoft Excel with MySQL Tables

1. Follow the instructions in the section "Adding a MySQL database as an ODBC User Data Source" for adding the desired MySQL database as an ODBC data source.

2. Open an Excel file. Click on "Data" from the top menu bar, and choose "**Import External Data**". Select "**New Database Query …**" from the sub-menu window. This will open the "**Choose Data Source**" window.

3. From the **Choose Data Source** window, select the desired DSN from the list, e.g., "MOVESDB", then click "**OK**".
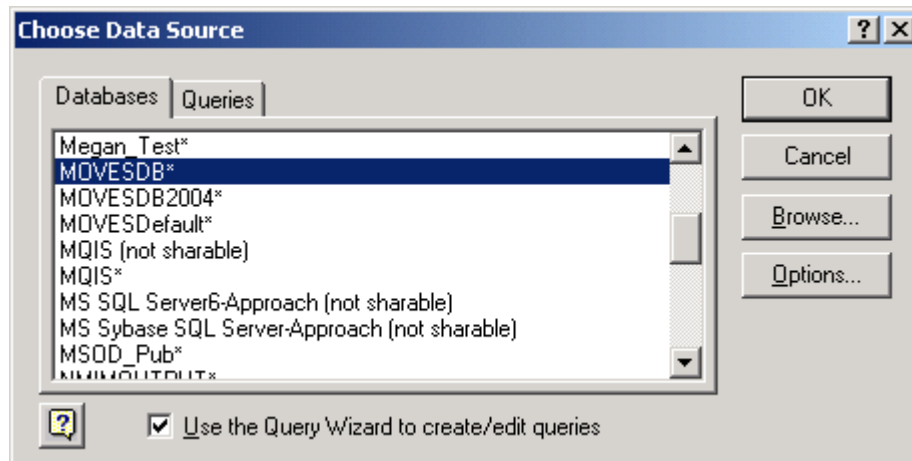
4. Click the "**OK**" button in the "**mysql Driver connect**" window.
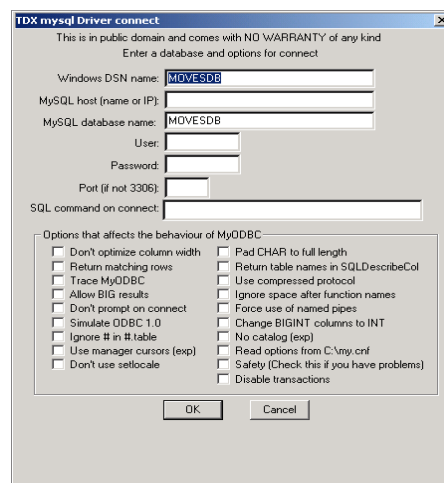
5. Follow the instructions posted by the Wizard until finished.

Appendix C - Using the Batch Command Line Interface

The MOVES command line allows MOVES to be executed without running its graphical user interface. It is useful in situations where repeated or unattended runs are needed, or for another computer program to execute MOVES. This interface presumes that a MOVES run specification file has been prepared and that the user is running from the DOS prompt.

These instructions presume some familiarity with DOS commands. The DOS command to execute the MOVES Command Line Interface is:

java MOVESCommandLine -r *runspecfile*

[OR]

java MOVESCommandLine -rl *runspeclistfile*

where "*runspecfile*" is the name of a file containing a saved MOVES run specification and "*runspeclistfile*" is the name of a text file containing a list of run specification file names, with one per line. The rest of the syntax is literal. The spelling of "MOVESCommandLine" is case sensitive because it is a java class name.

By executing one of these commands, the user runs the java interpreter (java.exe). MOVESCommandLine is a DOS parameter telling java.exe what java class file to begin executing, and the last two tokens are parameters passed to MOVESCommandLine.

Either version of this command can be executed from a DOS batch file, and batch files can contain multiple commands.

Prior to executing the command, the active directory should be set to the location where MOVES is installed, typically C:\Program Files\MOVES, and the SETENV.BAT file should be executed.

It can be difficult for DOS to find everything. For the command to work, three elements must be found:

- The java interpreter.
- The MOVESCommandLine java class.
- Any runspeclistfile and all runspecfiles.

Running SETENV.BAT insures that the java interpreter is found.

Running SETENV.BAT also insures that the procedure described in the next paragraph results in the MOVESCommandLine java class being found. Java experts can also use the CLASSPATH environment variable more directly to locate java class files.

If MOVES has been installed in the default location, C:\MOVES, then MOVESCommandLine.class is located at C:\MOVES\gov\epa\otaq\moves\master\commandline. This can be made the active directory, or the command can specify whatever part of the path is needed. For example, if the active directory is C:\MOVES, the command line interface class would be specified as "gov.epa.otaq.moves.master.commandline.MOVESCommandLine".

One way to insure that the runspecfile or the runspeclistfile is found is to specify the full path. If a simple file name is used the file should be located in the active DOS directory.

If the GUI is used to create an importer XML file (see Section 2.3.3.2), the XML file can be edited and executed via the command line. Remember that when using the command line, text output is not sent to the screen but instead stored in MOVESBatch.log. An example command line is:

java gov.epa.otaq.moves.master.commandline.MOVESCommandLine -e DEBUG
 -i importAllFromTemplates.xml

Be sure to place all of this on one line. The "-e DEBUG" portion ensures that all messages will be captured, even if they are just informational. The "-i" option directs MOVES to the XML file that describes the import actions to occur. Note that this XML file essentially contains a RunSpec within it, thus allowing wildcards even when importing via the command line.

Appendix D - Creating an On-Road Retrofit Input File

The "On-Road Retrofit" strategy, described in Section 2.9.9 allows the user to enter information about diesel trucks and buses that have been retrofit with emission control equipment. An On-Road Retrofit Parameters File must be input to the MOVES model to model on-road vehicle retrofit effects. This file can be in tab-delimited text, comma-delimited (*.csv), or Microsoft Excel (*.XLS) format. The user must create the Retrofit Parameters File by using either a text editor such as Wordpad or Notepad, or a spreadsheet such as Excel.

A. Header and Comment Rows

The first row of the Retrofit Parameter File must be a header row that contains the retrofit input parameter names separated by tabs. The remaining rows of the file contain the retrofit parameters. Comment rows are allowed if the first character in the row or the first cell in the tabbed file or *.XLS file is the '#' character.

B. Retrofit Parameter File Format

The Retrofit Parameter File must contain the following ten parameters in the order listed. Each parameter's possible values are provided.

1. Retrofit Pollutant

This column must contain the full MOVES pollutant name or the pollutant abbreviation for which the user wishes to model retrofit data. (In Draft MOVES2009 the user should not use the MOVES PollutantID number.) The data must appear exactly as shown in Table D-1 below, except that the pollutant abbreviation and/or full name input are not case sensitive.

Table D-1
Retrofit Parameter File Pollutants

PollutantID	Full MOVES Pollutant Name	Pollutant Abbreviation
1	Total Gaseous Hydrocarbons	HC
2	Carbon Monoxide (CO)	CO
3	Oxides of Nitrogen	NO_X
5	Methane (CH_4)	CH_4
6	Nitrous Oxide (N_2O)	N_2O
90	Atmospheric CO_2	AT CO_2
91	Total Energy Consumption	TotEnergy
92	Petroleum Energy Consumption	PetEnergy

| 93 | Fossil Fuel Energy Consumption | FossilEnergy |

(cont.)

Table D-1
Retrofit Parameter File Pollutants (cont.)

PollutantID	Full MOVES Pollutant Name	Pollutant Abbreviation
98	CO_2 Equivalent	CO2EQ
101	Primary PM10 - Organic Carbon	PM10OC
102	Primary PM10 - Elemental Carbon	PM10EC
105	Primary PM10 - Sulfate Particulate	PM10Sulfate
106	Primary PM10 - Brakewear Particulate	PM10Brake
107	Primary PM10 - Tirewear Particulate	PM10Tire
111	Primary PM2.5 - Organic Carbon	PM25OC
112	Primary PM2.5 - Elemental Carbon	PM25EC
115	Primary PM2.5 - Sulfate Particulate	PM25Sulfate
116	Primary PM2.5 - Brakewear Particulate	PM25Brake
117	Primary PM2.5 - Tirewear Particulate	PM25Tire

2. Retrofit Emission Process Input

This column must contain the full MOVES process name or the ProcessID abbreviation for which the user wishes to model retrofit data. In Draft MOVES2009, the user should not use MOVES ProcessID. The data must appear exactly as shown in Table D-2 below, except that the ProcessID abbreviation and full name are not case sensitive.

Table D-2
Retrofit Parameter File Processes

ProcessID	Full MOVES Process Name	Process Abbreviation
1	Running Exhaust	Running
2	Start Exhaust	Start
90	Extended Idle Exhaust	Extended Idle
9	Brakewear	Brake
10	Tirewear	Tire

3. Retrofit Fuel Type Input

This column must contain the full MOVES fuel name or the fuel abbreviation for which the user wishes to model retrofit data. In Draft MOVES2009, the user should not use FuelTypeID. The data must appear exactly as shown in Table D-3 below, except that the fuel abbreviation and full name are not case sensitive.

Table D-3
Retrofit Parameter File Fuel Types

FuelTypeID	Full MOVES Fuel Type Name	Fuel Type Abbreviation
1	Gasoline	Gas
2	Diesel Fuel	Diesel
5	Ethanol (E85)	Ethanol

4. Retrofit Vehicle Source Type Input

This column must contain the Sourcetype abbreviation or the full MOVES Sourcetype name for which the user wishes to model retrofit data. In Draft MOVES2009, the user should not use SourceTypeID number. The data must appear exactly as shown in Table D-4 below, except that the sourcetype abbreviation and full name input are not case sensitive.

In Draft MOVES2009, only diesel (FuelTypeID = 2) vehicles with the following SourceTypeIDs are covered by the Retrofit Calculations: 32, 41, 42, 43, 51, 52, 53, 61, and 62.

Table D-4
Retrofit Parameter File Source Types

SourceTypeID	Full MOVES Vehicle SourceType Name	SourceType Abbreviation
11	Motorcycle*	MC
21	Passenger Car*	Car
31	Passenger Truck*	PTruck
32	Light Commercial Truck	ComTruck
51	Refuse Truck	RefuseTruck
52	Single Unit Short-Haul Truck	SUShortTruck
53	Single Unit Long-Haul Truck	SULongTruck
54	Motor Home	MH
43	School Bus	SBus
42	Transit Bus	TBus
41	Intercity Bus	IBus
61	Combination Short-haul Truck	CShortTruck
62	Combination Long-haul Truck	CLongTruck

* These sourcetypes should not be used in retrofit input files.

MOVES compares the user inputs for pollutant, process, fuel type, and source type with the stored retrofit possibilities and prompts the user if an incorrect name, abbreviation or number for pollutant, process, fuel type, or source type is entered.

5. Initial Calendar Year of Retrofit Implementation

The Initial Calendar Year of the Retrofit Implementation is the first calendar year that a retrofit program is administered. Initial Calendar Year input must be equal to or less than the Final Calendar Year of Retrofit Implementation. All months within a calendar year are affected equally by the retrofit.

6. Final Calendar Year of Retrofit Implementation

The Final Calendar Year of the Retrofit Implementation is the last calendar year that a retrofit program is administered. Final Calendar Year input must be equal to or greater than the Initial Calendar Year of Retrofit Implementation.

7. Initial Model Year that will be Retrofit

The Initial Model Year that will be Retrofit is the first model year of coverage for a particular vehicle class/pollutant combination. Valid entries for initial model year must meet the following mathematical requirement:

Initial Model Year ≥ Initial Calendar Year - 30

The Initial Model Year cannot be greater than the Final Model Year that will be Retrofit.

8. Final Model Year that will be Retrofit

The Final Model Year that will be retrofit is the last model year of coverage for a particular vehicle class/pollutant combination. No retrofit will be performed on Final Model Year input which is larger than the Evaluation Calendar Year. Also, the Final Model Year input cannot be less than the Initial Model Year that will be Retrofit.

9. Percentage of the Fleet Retrofit per Year

The Percentage of the Fleet Retrofit per Year represents the percentage of VMT of a particular fleet of a particular vehicle class, retrofit calendar year group, model year group, and pollutant combination that is to be rebuilt in a given calendar year. For a successful retrofit simulation, a value greater than zero and less than or equal to 100.0% must be entered. MOVES checks to ensure that the product of the number of calendar years of retrofit coverage (Final Calendar Year of Retrofit Implementation - Initial Calendar Year of Retrofit Implementation) and the Percentage of the Fleet Retrofit per Year does not exceed 100%. For example, a retrofit simulation will be flagged as invalid, and an error message will appear, if the simulation has a retrofit program start in calendar year 2005, a program end in calendar year 2008, and a yearly Fleet Retrofit Percentage of 50% (3 * 50% > 100%).

10. Percentage Effectiveness of the Retrofit

The Percentage Effectiveness of the Retrofit is the percent emission reduction achieved by a retrofit. It is computed from a non-retrofit emission baseline. The user must enter a retrofit effectiveness value for a particular vehicle class, retrofit calendar year group, model year group, and pollutant combination. All values up to 100% are valid. A negative value is permitted because it implies an emission increase as a result of retrofit, which can occur. A value greater than 100% is not permitted because it implies negative emissions will be generated.

Appendix E - Best Practices

There are several "best practices" that can assist the user in managing various aspects of the MOVES process. In particular, we have found it helpful to be careful and consistent in naming, managing & documenting RunSpecs and files.

Table E-1
MOVES Best Practices

Suggestion	Rationale	Example
Name all your MOVES output databases with a code indicating that the database includes MOVES output	Prevents overwriting input files; makes it easier to identify output files.	Washtenaw2008_MO (for MOVES Output) or just Washtenaw2008_out
When testing MOVES, name output with "Temp" or other indicator.	Mysql/data folder can get cluttered. This allows you to clean up more easily	Washtenaw2008_temp_MO
Name all your MOVES input databases and strategy files with a code indicating that the database contains MOVES alternate input data.	Helps you organize your inputs and prevents you from inadvertently damaging important input information.	Washtenaw2008_IN
Name all your MOVES run specification text files with a code indicating that the file contains a MOVES run spec.	Helps you organize your input information and can be used to locate files using a search.	Washtenaw2008.mrs (for MOVES Run Specification) or Washtenaw2008_mrs.txt
Name all the files you use for a specific run using the same name, but with extensions to identify the parts (see file naming suggestions above).	Helps you organize your information.	Washtenaw2008_in, Washtenaw2008.mrs, Washtenaw2008_mo

Suggestion	Rationale	Example
Keep all of your run specifications in the same directory (RunSpecs). This folder should be outside the folders installed by MOVES so that it will not be overwritten when installing a new version of MOVES.	MOVES will always look in the last location you accessed, so keeping them all in one place keeps you from searching for your runspecs.	My Documents\RunSpecs
Decide on the Scale you wish to use before making selections on any other panels.	Changing the Scale option after making choices in other panels may require you to re-do those panels.	
Using the description panel is helpful when trying to discriminate between several runs. Think about filling out some sort of repeatable formula such as date, relevant important parameters, who is doing the run, etc. Note that MOVES preserves these descriptions in the movesrun output table for future reference.	Documentation of the RunSpec is very important, especially if many scenarios are being processed.	
Every run used for any official purpose should be archived completely. All batch files, RunSpecs, MOVES code and configuration files, supplemental code copied into MOVES by batch files, default database, user input databases, etc. should be saved.	Months or years later, the run can be reproduced by anyone opening the archive.	

Suggestion	Rationale	Example
If they are not too large, output related runs should be saved to the same MySQL database	Combined with useful Run descriptions, you can keep all the data, and its description in a single place. Looking at the Moves Run table in that database can be very informative.	

Appendix F - Scenario 1: Estimating Changes in CO2 using the AVFT

Explanation of Scenario 1: Estimating Changes in CO2 using the AVFT

This AVFT Scenario.mrs RunSpec (found in the User Guide Example files\AVFT Example folder on the MOVES Installation disk) demonstrates how the EPA's Motor Vehicle Emission Simulator (MOVES) could be used to generate estimates of light-duty vehicle greenhouse gas (GHG) emissions and fuel use. The "policy scenario" discussed here consists of potential increases in car and truck CO_2 standards, while the baseline scenario is similar to the MOVES defaults. To model these scenarios, we created new MOVES inputs to simulate potential vehicle CO_2 limits.

Creating new MOVES inputs involved multiple steps. First, Alternative Vehicle Fuels and Technologies (AVFT) files were created to model the CO_2 limits. These files are used to specify the fraction of the vehicle fleet in a given calendar year that is comprised of vehicles of a certain vehicle technology (e.g., conventional gasoline, conventional diesel, electric vehicle, etc.). For simplicity, CO_2 limits were simulated by shifting a fraction of the gasoline and diesel vehicle fleets into the electric vehicle (EV) fleet. Since EVs do not contribute to onroad CO_2 emissions, shifting vehicles to the EV fleet has the same effect as improving the efficiency of the gasoline and diesel fleets in terms of total CO_2 emissions. It should be noted this approach is not predicting that EVs will be used to meet tighter CO_2 limits, it is simply a method used to achieve the desired overall fleet total CO_2 levels.

Use of the AVFT:

This example uses the MOVES Alternate Vehicle Fuels and Technologies (AVFT) Strategy tool to estimate the effect of a change in fuel economy standards. Incorporating potential CO_2 limits into the MOVES AVFT files was achieved by shifting a fraction of the gasoline and diesel fleets into the electric vehicle fleet to simulate increased fleetwide energy efficiency relative to the MOVES default assumptions.   The basis of the MOVES CO_2 estimates is energy consumed. We can model new potential CO_2 limits by reducing energy consumption rates in MOVES using fuel consumption as a surrogate for these reductions; hence our discussion focuses on MPG.

Table II-1 shows the fuel economy assumptions for cars in the baseline and policy scenarios. These fuel economy values are unadjusted values (i.e. they are on the same basis as CAFE ratings), so they are higher than the actual onroad fuel economy; CAFE values were used to generate the percent reductions in fuel consumption that were used to calculate the on-road emissions calculated by MOVES. As MOVES simulates on-road fuel economy, the values output by the model will be lower than those in Table II-1

Table II-1
MPG values for Baseline and Policy Scenarios

Model Year	Cars	
	baseline	*Policy Scenario*
2001	29.8	29.8
2002	29.8	29.8
2003	29.8	29.8
2004	29.8	29.8
2005	29.8	29.8
2006	29.8	29.8
2007	29.8	29.8
2008	29.6	29.6
2009	29.7	29.7
2010	29.7	29.7
2011	30.1	30.1
2012	30.4	30.4
2013	30.7	30.9
2014	30.7	32.2

The diesel market share was assumed to be the same for both scenarios, as shown in Table II-2.

Table II-2
Diesel market share (%)

Model year	Cars
2001 - 2010	1.0
2011	1.2
2012	1.3
2013	1.3
2014	1.3

For a given model year, the percentage of electric vehicles was calculated based on the scenario fuel economy, and the diesel market share, as follows:

$$\text{Scenario} \% EV = \left[1 - \left(\frac{\frac{1}{ScenarioFuelEconomy}}{\frac{1}{DefaultFuelEconomy}} \right) \right],$$

For example, the percentage of EVs for cars in MY 2014, for the policy scenario, was:

$$\text{Scenario} \% EV = \left[1 - \left(\frac{\frac{1}{32.2}}{\frac{1}{30.7}} \right) \right] = 4.66\% ,$$

The percentages of gasoline and diesel vehicles in the car and truck fleets were then calculated such that the percentages of diesels in the car and truck fleets, excluding the EVs, equaled the market share percentages specified in Table II-2, as follows:

$$\% diesel = \left(1 - \% EV \right) \times DieselMarketShare$$
$$\% gasoline = (1 - \% EV) \times (1 - DieselMarketShare)$$

For example, returning to cars in MY 2014, Policy scenario, the gasoline and diesel percentages were:

%diesel   = (1-0.047) x .013 = 1.24%
%gasoline   = (1- 0.047) x (1-0.013) = 94.10%

The appropriate percentages were entered into the AVFT Scenario.mrs RunSpec using the AVFT. When the AVFT Scenario.mrs RunSpec is loaded into MOVES, the values can be viewed using the MOVES GUI AVFT Strategy panel.

Running Scenario 1

To run Scenario 1, start up the MOVES program, then use the File menu to Open the AVFT Scenario.mrs RunSpec. This RunSpec includes the AVFT information described above. Go to the General Output panel to create an output database named "Scenario1_mo". Use the File Menu command to save the RunSpec, then use the Action menu to Execute the run. The run will take between 10 and 30 minutes on most computers.

Analysis of Results

After running Scenario 1, use MySQL Query Browser or other database tools to examine the output database.

1. You can check to see that the electric vehicles use no on-road energy.

SELECT * FROM movesoutput m where yearid=2014 and modelyearid=2014 and sourcetypeid=21 and pollutantid=90;

Look at the emissionquant column, and match it to the various fueltypes. Notice that fuel type 9 = 0 for both starts and running, while fuel types 1 and 2 have emission output.

2. Go to the movesoutput table, and select pollutant id for CO_2 (90) along with the start and running processes (1 and 2). Sum the emission quantity (emissionquant) and record this value.

   SELECT sum(emissionquant) FROM movesoutput m where yearid=2014 and modelyearid=2014 and sourcetypeid=21 and pollutantid=90 and fueltypeid in (1,2);

3. Look at the movesactivityoutput table and in the year 2014, sum the VMT from the cars with the modelyear 2014. Record this value.

   SELECT sum(activity) FROM movesactivityoutput m where yearid=2014 and modelyearid=2014 and sourcetypeid=21;

   By including the electric fueled mileage in the activity output, you can simulate a CO_2 standard for cars which is higher than the MOVES default. Dividing the emission output by the activity provides grams of CO2 per mile. This number can be compared against a similar run without an AVFT in order to compare the potential CO_2 improvements over a baseline scenario. Remember, the MOVES output is based on-road fuel consumption, which is higher than indicated by CAFE standards.

Appendix G - Scenario 2: MOVES Project Level Example

1. Definition of the MOVES Project

       In this example MOVES Runspec, the EPA's Motor Vehicle Emission Simulator (MOVES) was used to generate emission inventory estimates from a hypothetical scenario where heavy-duty vehicle traffic is entering and departing a parking lot using a single roadway. Since this example is only for illustration of the MOVES software features, it was kept intentionally simple. It does <u>not</u> represent any specific real-world project.

       The Project constraints are:

1.   There are only two links (inbound and outbound from a parking lot)
2.   The two links and the off-network link are independent of each other.
3.   A simple off-network link contains all of the parking, extended idle and vehicle start operations.
4.   Only heavy-duty vehicles operate on the roadway and are present in the off-network area.
5.   The project example only models nitrogen oxide (NOx).
6.   One hour of operation was selected. If the user desired to model additional hours, then additional MOVES Projects need to be created.

       Figure 1   Basic Schematic of the Project
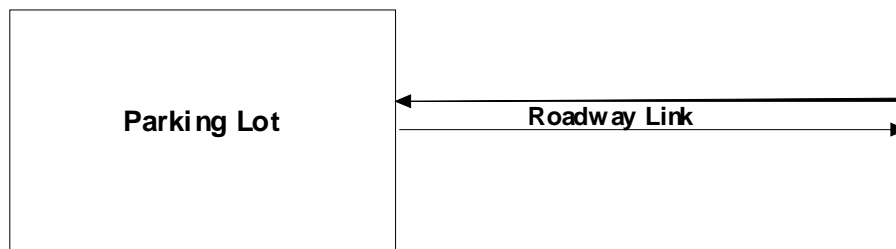

       Figure 1 shows a basic schematic of the Project to be modeled. It consists of a parking lot and a two-way road link leading in and out of the parking lot. The specific modeling parameters are shown and discussed in Tables 1, 2 and 3 below.

Table 1
Summary of On-road Project Level Parameters

Location County	Washtenaw County, Michigan
Calendar Year	2009
Month	January
Time	11:00 PM to 11:59:59 PM (hour 24)
Weekday/Weekend	Weekday
Temperature	28.5 F
Humidity	69.5 %
Road type of links (link1 and link 2)	Rural Unrestricted Access – can represent any non freeway rural road
Roadway Link Length (both links)	0.998 miles
Link Traffic Volume – In (link 1)	100 vehicles per hour
Link Traffic Volume – Out (link 2)	10 vehicles per hour
Link Average Speed – In (link 1)	28.5 miles per hour
Link Average Speed – Out (link 2)	28.5 miles per hour

Only one county may be chosen for a given project level run. In this example, Washtenaw County, Michigan was chosen. Any of the 3,222 counties or a generic county (defined by the user) may be chosen. Only one calendar year, one month and one hour may be chosen for a given project level run. If the user desires to model more than one such entity (i.e., all 24 hours), then they must perform additional and separate project level runs. It is suggested for such runs that the user employ MOVES batch mode features. A project level run may include either weekday or weekend or both. In this example only weekday was included. A temperature in degrees Fahrenheit (28.5 F), and relative humidity of (69.5 percent) were also used. These are the default MOVES values for this county, month and hour. However, any reasonable values may be entered for these parameters.

For each roadway link the user must specify a MOVES road type which best represents it. In this example, the rural unrestricted access road type was chosen to represent both links. The user must also specify a roadway link length for each link they wish to model. In this case because it is the same road in and out of the parking lot, both links are 0.998 miles in length.

The traffic volume must also be specified for each link. This is the total average traffic flow from all vehicle types on the link during the hour period.   In this simple example, all of the vehicles are heavy-duty trucks, but in general any and all of the MOVES vehicle source types may be included at the same time in a project level run. The distribution of the traffic by MOVES source type is an additional input.

The average speed on each link must also be specified. This is used for internal calculations in the project level model. If the user provides a driving schedule input [speed / grade inputs versus time (seconds)], then the driving schedule input and the average speed input need to be consistent. In this case, a driving schedule input is provided for both links. Figure 1 shows the driving schedule in graphical form. Link 1 shown in blue, has driving starting at 30 mph and decreasing to zero mph as the vehicles enter the parking lot. Link 2 shown in green has

driving starting at zero mph and accelerating to a 30 mph cruise. The driving schedule data, and all the rest of the required data for the project level example, is contained in the input spreadsheet "Project Level HD Example.xls"

Figure 1

Speed / Time Trace for Example Links
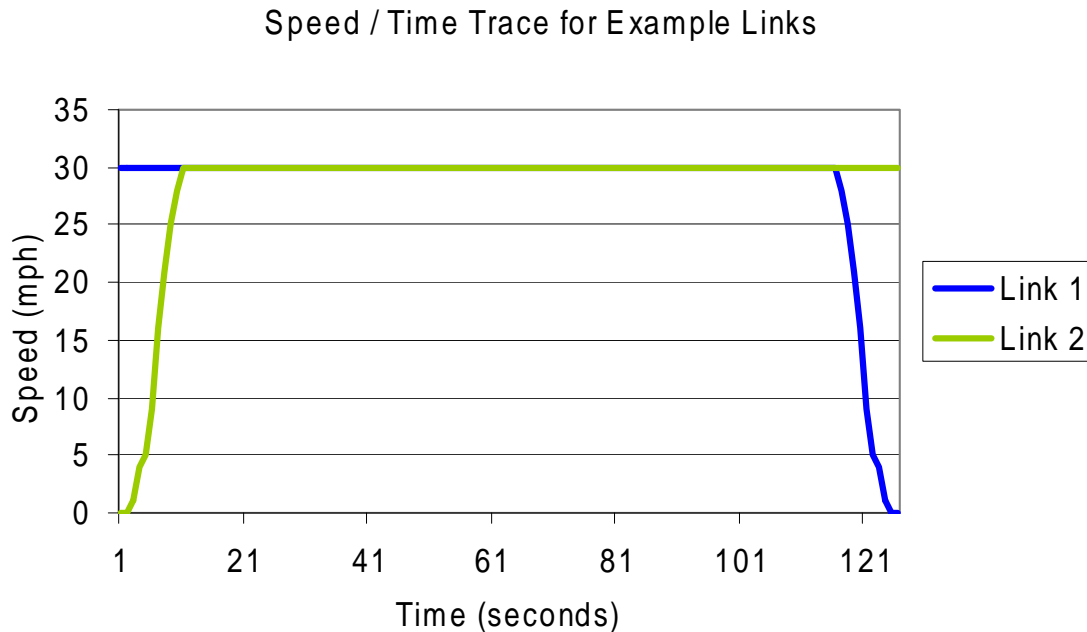
For this example, the off-network parameters are shown in Table 2. These four parameters include a vehicle population of 200 vehicles in the parking lot <u>on average</u> during the hour period. This input is an average value, because some vehicles may have been in the lot at all times while others entered or exiting during the hour period. In this example, the start fraction is set to three percent (0.03). This is the fraction of the average vehicle population which started the vehicle during the hour. If all of the vehicles in the lot on average had been started, then this value would be unity (1.0). It may also be greater than unity if large numbers vehicles are repeatedly started during the period.

The extended idle and parked vehicle fraction parameters were set to 0.90 and 0.09 respectively, for this example. The 90 percent input for extended idle reflects the fact that 90 percent of the total vehicle – hours (only one hour by definition) in the parking lot were spent in extended idle mode (vehicles are parked in a lot with their engines idling at higher than curb idle speeds). The 9 percent input for parking reflects the fact that 9 percent of the total vehicle – hours in the parking lot were spent in park mode (vehicle is parked and the engine is off). The sum of extended idling and parking cannot be greater than unity.

Table 2
Summary of Off-Network Project Level Parameters

Average Vehicle Population	200 vehicles in the parking lot on average
Start Fraction	0.03
Extended Idle Fraction	0.90
Parked Vehicle Fraction	0.09

The operating mode distribution for the parking lot link must be entered. This is a distribution by AVERAGE vehicle 'soak' time (i.e., the time since a vehicle was last started). For example, the table below shows that 5 percent of the vehicles in the parking lot have not been started for more than 720 minutes or 12 hours. Extended Idle operating model fraction is always set to unity (1.00) when it is present because it is the only operating mode of its type. This input should not be confused with the off network parameter called Extended Idle Fraction with a value of 0.90 that is shown in Table 2.

Table 3
Operating Mode Distribution Parameters for Start and
Extended Idle Processes

OpmodeID Code	Operating Mode Description	Operating Mode Fraction
101	Soak Time < 6 minutes	0.00
102	6 minutes <= Soak Time < 30 minutes	0.05
103	30 minutes <= Soak Time < 60 minutes	0.30
104	60 minutes <= Soak Time < 90 minutes	0.10
105	90 minutes <= Soak Time < 120 minutes	0.50
106	120 minutes <= Soak Time < 360 minutes	0.00
107	360 minutes <= Soak Time < 720 minutes	0.00
108	720 minutes <= Soak Time	0.05
200	Extended Idle Operating Mode	1.00

Table 4 shows the source type age distribution for the vehicles in the project. In this example, there is only one source type (heavy-duty long haul trucks) present. Thus, only one age distribution is required. Additional age distributions would be required if additional source types were present. The age distribution runs from age zero (brand new) to 30 years. All ages greater than 30 years are included in the age 30 group. The distribution must sum to unity within a source type. **Note that** the project level model does not allow separate age distributions for different fuel types if the same source type is selected. For example, the same age distribution would be used for both gasoline and diesel long haul trucks if both were present in the project.

Table 4
Source Type Age Distribution

Source Type	ageID	ageFraction
62	0	0.2
62	1	0.15
62	2	0.1
62	3	0.1
62	4	0.1
62	5	0.07
62	6	0.05
62	7	0.05
62	8	0.05
62	9	0.02
62	10	0.02
62	11	0.01
62	12	0.01
62	13	0.01
62	14	0.01
62	15	0.01
62	16	0.01
62	17	0.01
62	18	0.01
62	19	0.01
62	20	0
62	21	0
62	22	0
62	23	0
62	24	0
62	25	0
62	26	0
62	27	0
62	28	0
62	29	0
62	30	0

2. Example Data Creation and Input

All inputs for the example MOVES Project Level run are in the Excel Workbook - **Project_Level_HD_Example.xls**.

The inputs are in the spreadsheets.

1.   DriveScheduleSecondLink
2.   OffNetworkLink
3.   ZoneMonthHour
4.   LinkSourceTypeHour
5.   SourcetypeAgeDistribution
6.   FuelSupply
7.   Link
8.   OpModeDistribution

        In this example, all of the inputs for the Project Level were read from a single Excel workbook. However, it is recommended that the user use the *Create Template* button for each of the Project Level input tabs and create a set of Excel workbook templates (with a set of individual worksheets - one for each input tab) and modify / populate these templates with the actual data. The spreadsheets within each template will provide necessary descriptions of the data fields and moves codes (i.e., countyID codes, fuelformulationID codes, roadtypeID codes, hourDayID codes and sourcetypeID codes. The MOVES project level importer will show a list of all of the individual Excel worksheets from an Excel workbook. The user must pick the 'active' worksheet (usually the leftmost worksheet in the workbook or the first spreadsheet entry in the MOVES importer) as an input

        The user should also note that the DriveScheduleSecondLink and the opmodeDistribution table need not exist together for all of the roadtype links. In this example, the DriveScheduleSecondLink table contains an average speed-time 'trace' for both roadway links. It is used by MOVES to internally create an opmodeDistribution for each roadway link. A separate opmodeDistribution for the road links is not required to be supplied, but could have been supplied in-lieu of the DriveScheduleSecondLink data input. The opmodeDistribution is always required for the start operation parameters (table 3 contains the inputs). It contains the opmodeDistribution for the NOx emission starts. These values are used to differentiate start soak times.

        The first input step is to create the Project Level database where the imported data is stored. This is a MySQL database and it is named "projectHDinput" in this example.

        The user loads the data into the Project Level database (i.e., projectHDinput) for each input tab by browsing for the individual file (use the Browse button), and once found, pressing the "Import" button to import the data into MOVES. A message diagnostic of "Import Complete" will be issued if the data import was successful. This process is repeated for each of the tabs.

        After the data loading process is complete the user should press the *Done* button to save the data in the database. This will exit the Project Level data manager. As a final step, the user

may be required to go to the Geographic Panel tab and choose the new Project Level database **("projectHDinput")** from a list of databases. If the new database is not visible, the user should press the *Refresh* button.

Both the Excel input file **Project_Level_HD_Example.xls** and the MOVES project level input database **projectHDinput** are provided in the MOVES package as an assistance to the user.

3. MOVES Runspec

A MOVES Runspec **(called Project_HDExample.mrs)** was created to model this example MOVES Project level analysis. A standard MOVES Runspec was created first. The user should note that a Project level run MUST contain only ONE

> County
> Year
> Month
> Hour

The example Runspec was further simplified to include only one sourcetype, one roadtype, one weekday/weekend combination and three pollutant / process combinations. The runspec is provided as an example and the user should load it to become more familiar with this example.

4.   MOVES Simulator Results for the Example

The results for this simple MOVES Project Level simulation are reported in the MySQL database "**ProjectLevelHDOutput**". They are summarized below.

table

Table 5
MOVES Project Level Example Simulation Results

LinkID	Link Description	NOx emissions (grams)
1	Inbound Road	1154.582
2	Outbound Road	129.330
3	Parking Lot (extended idle)	8349.843
3	Parking Lot (start)	3.574

The emissions are reported as an 'inventory' for the project and are broken down by linkID (roadway and off-network – the user must specify that the results to be reported by roadway and process in the Runspec). In this example, the result are reported in units of grams of NOx emissions. The time domain for the Project Level is always one hour. The user should also remember that these results are **<u>average</u>** results over the one hour time domain and the geographic link domain. One reason project level reports 'average' results is because the

model's calculation methodology assumes (for simplicity) independence of the individual links in the project, and does not dynamically calculate traffic flows between links or residence times in off network (the user must do this step **before** the data is entered into the MOVE project level simulator.

The large difference in NOx emission inventories between two physically similar links (link 1 and link 2) is because of the large difference in traffic volumes. The traffic volume for link 1 is an order of magnitude greater than the traffic volume for link 2. The relatively small amount of NOx emissions from the start process is the result of a very low start fraction input (i.e., 3 percent). The relatively large amount of emissions from the extended idle process reflects a fairly high fraction of extended idle operation (i.e., 90 percent). In comparison, the running operation of the inbound road contains half as many vehicles (100 vehicles versus 200 in extended idle), and the running operation on average lasts only about 2 minutes per vehicle versus the entire hour for 90 percent of the vehicles.

Appendix H - Scenario 3: County Data Manager and I/M Editor Example

1. Explanation of Scenario 3

MOVES becomes an official EPA model, the County Data Manager and I/M Editor are expected to be used extensively by users, particularly by nonattainment or maintenance areas when conducting SIP or conformity analyses. These tools provide the user the ability to import and edit local data, rather than using MOVES defaults.

In this scenario, the County Data Manager and I/M Editor are used to construct a RunSpec using area-specific data. The area being modeled will be a custom domain. While there are some differences between using the custom domain option and selecting an individual county, the basic operation of the County Manager and I/M Editor are the same. Where differences exist, this scenario will describe how a custom domain and a single county would be treated differently.

2. Constructing the RunSpec

The custom domain in this scenario will be named "Example City" and will be an ozone nonattainment area, so the selections and use of the County Data Manager and I/M Editor will reflect that assumption (i.e. on the **Pollutants and Processes** panel, Running Exhaust for Total Gaseous Hydrocarbons and Oxides of Nitrogen and Evap Fuel Vapor Venting for Total Gaseous Hydrocarbons were selected because these pollutants are associated with the formation of ground-level ozone and are affected by I/M). The RunSpec should be filled out in its entirety before either the County Data Manager or the I/M Editor are used. In this example, some selections were limited to reduce the run-time of the RunSpec (e.g. only Passenger Cars were selected on the **On Road Vehicle Equipment** panel).

Open the file name "examplecity.mrs" (located in the USER Guide Example Files\City Example\City Example Files folder) to view the RunSpec and to create an output database using the General output panel (e.g. examplecity_output). Then select the RunSpec Geographic Bounds panel and click the "Enter/Edit Data" button. This will open the County Data Manager window.

3. Using the County Data Manager

With the County Data Manager open, name and create a user-input database (e.g. examplecity_import). Next, files containing the area-specific data must be imported on each tab. The importing process is the same for all the tabs, so it is explained generally in the next paragraph.

The Excel files containing the example area-specific data are included as part of the MOVES installation pack and have the following naming convention: "examplecity_[TABNAME]". Importing the files is straightforward - simply click the "Browse" button, find where the file is saved, and select the appropriate file for the tab (i.e. if working in

the "Age Distribution" tab, select the file "examplecity_agedistribution.xls"). After the file has been selected, a small pop-up window appears:
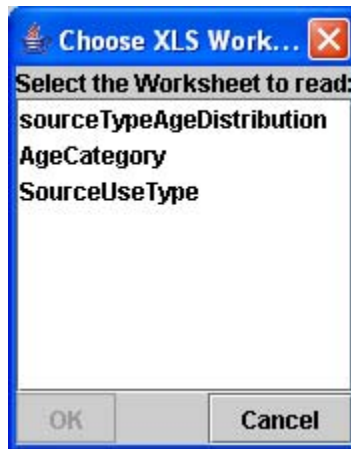
Select the correct worksheet (in this example, it will be the first worksheet listed and will have a name similar to the tab name) and click "OK."
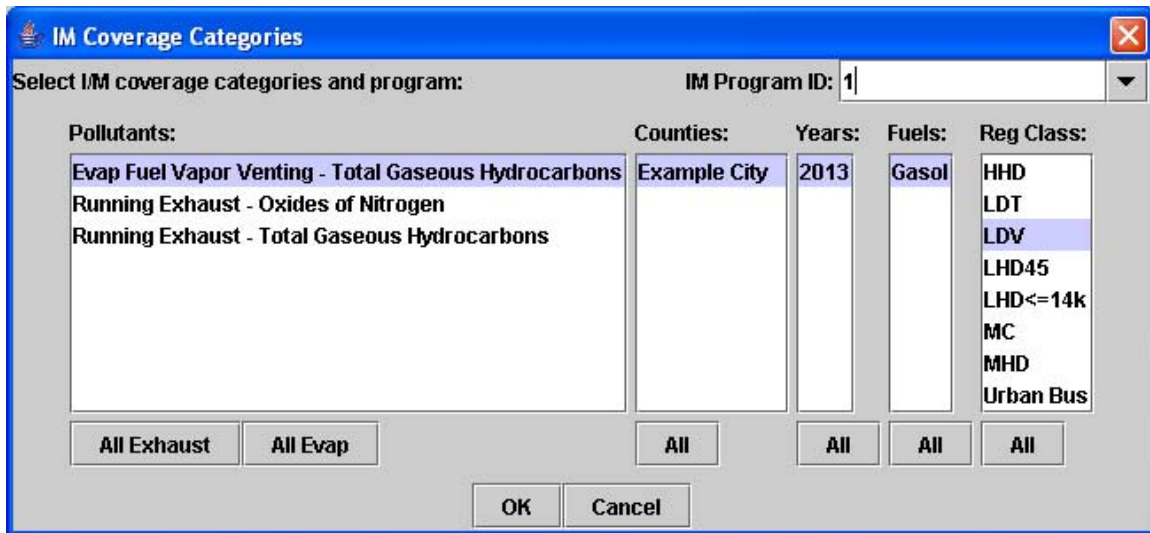
Next click the "Import" button. The "Messages" box should say, "Import Complete." If there were problems with the import, error messages here would describe the problem.   In case of error, the user should revise the input, then click, the "Clear Imported Data" button before reattempting the import.

        The County Domain Manager can be used to input data for a specific county or to create a custom domain. Currently, the only difference is that the custom domain option includes a "Zone Road Activity" tab and users are required to fill-in a value of 1 for all rows in "SHOAllocFactor" column. This tab exists to allow the Source Hours Operating (SHO) to be allocated to different zones within a custom domain; however, this capability is not fully implemented in DRAFT MOVES2009.. It is possible to create multiple zones using the **Manage Input Data Sets** panel, but this feature is not illustrated here.

4. Using the I/M Editor

        After data have been imported in all of the tabs in the County Data Manager (all the tabs should have green checks), the user can go to the "IM" tab and click the "Display/Edit IM Program Coverage Records" button to open the I/M Editor. Because the custom domain option is being used, there are no default I/M programs; however, if a county is selected, some default I/M programs may exist and be listed.

        Since no default I/M programs exist in the custom domain option, the "Generate New and Replacement Records" button should be clicked, which will open the **IM Coverage Categories** panel.
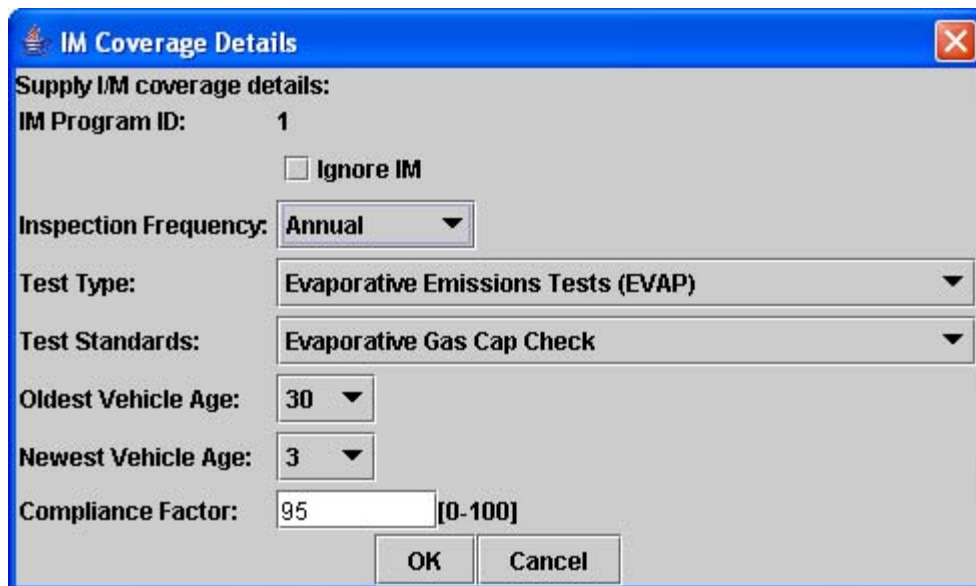
The panel should be modifed as illustrated above to create a new I/M program. The user should enter an IM Program ID of "1" and should select, the Pollutant "Evap Fuel Vapor Venting - Total Gaseous Hydrocarbons" and the Reg Class "LDV." (Note that "Example City", "2013", and "Gasol" are selected automatically because these are the only possible selections in these columns.) Once these entries have been made, click "OK," and the "IM Coverage Details" window will open:

The user should modify the IM Coverage Details panel as illustrated above. This process could be repeated to create additional I/M programs for the exhaust pollutants, or the additional I/M programs could be imported into the I/M Editor by simply clicking the "Import" button, finding and selecting the file "examplecity_im.xls", and choosing the "IMActive" worksheet.   The resulting IM coverage is displayed in the IM Coverage in MOVES panel, as illustrated below:

♀ **Note**; There are several additional worksheets included in the "examplecity_im.xls" file to provide examples of how changes can be made to the I/M programs. These changes can also be made by clicking the "Edit Selected Records", "Generate New and Replacement Records", or "Ignore I/M"/ "Use I/M" buttons; however, if using the "Edit Selected Records" button, only one record (or extremely similar records) should be selected to prevent illogical process/test type combinations (e.g. "Evap Fuel Vent" process, but "Exhaust OBD" test type).

The custom domain user-input database process has been completed. Click "Exit" in the I/M Editor and "Done" in the County Data Manager to return to the **Geographic Bounds** panel. Make sure the user-input database is selected in the **Domain Input Database** section of the **Geographic Bounds** panel.

5. Executing the RunSpec and Analyzing the Results

The RunSpec and county-input database have both been completed, so the RunSpec can now be executed by selecting **Execute** from the **Action** drop-down menu.

The results for this simple MOVES County Data Manager Example are reported in the MySQL database "examplecity_output". They are summarized below.

Table H-1 Example City - July 2013 Weekday Emissions from Passenger Cars

Pollutant (pollutantID)	Process (processID)	Emissions (grams)
Total Gaseous Hydrocarbons (1)	Running Exhaust (1)	943643
Total Gaseous Hydrocarbons (1)	Evaporative Fuel Vapor Venting (12)	748853
Oxides of Nitrogen (3)	Running Exhaust (1)	3913423