Motor Vehicle Emission Simulator (MOVES) 2010

User Guide

United States
Environmental Protection
Agency

Motor Vehicle Emission Simulator (MOVES) 2010

User Guide

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

United States Environmental Protection Agency

EPA-420-B-09-041
December 2009

Table of Contents

Page

1.0. Getting Started with MOVES

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

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

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

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

.

1.1 About MOVES

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

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

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

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

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

1.2 MOVES Documentation

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

1.2.1 About this Manual

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

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

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

NOTE Notes contain important information about the panel being described.

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

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

1.2.2 Other Documentation and Online Help

Additional documentation covering the following topics is available for MOVES2010.

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

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

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

1.2.3 What's Next? Working with MOVES

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

⇒ If you have just installed MOVES or are uncertain about your installation, follow instructions in "Testing Your Installation," provided in Section 1.3.2 of this guide.

⇒ If MOVES is installed and operational on your computer, you should run the example to become more familiar with the tool before moving on. See "Executing Example MOVES2010 Run Specification" included in Section 1.4.

⇒ If you would like to become more familiar with MOVES terminology and design, see the "MOVES Software Design and Reference Manual," available in .pdf format on the MOVES website.

⇒ If you have questions about the MOVES Graphical User Interface (GUI) or executing a run, see "Overview of MOVES RunSpec User Interface" in Section 2 of this guide**.**

⇒ If you are ready to begin using MOVES, see "Starting MOVES," also in Section 2 of this guide.

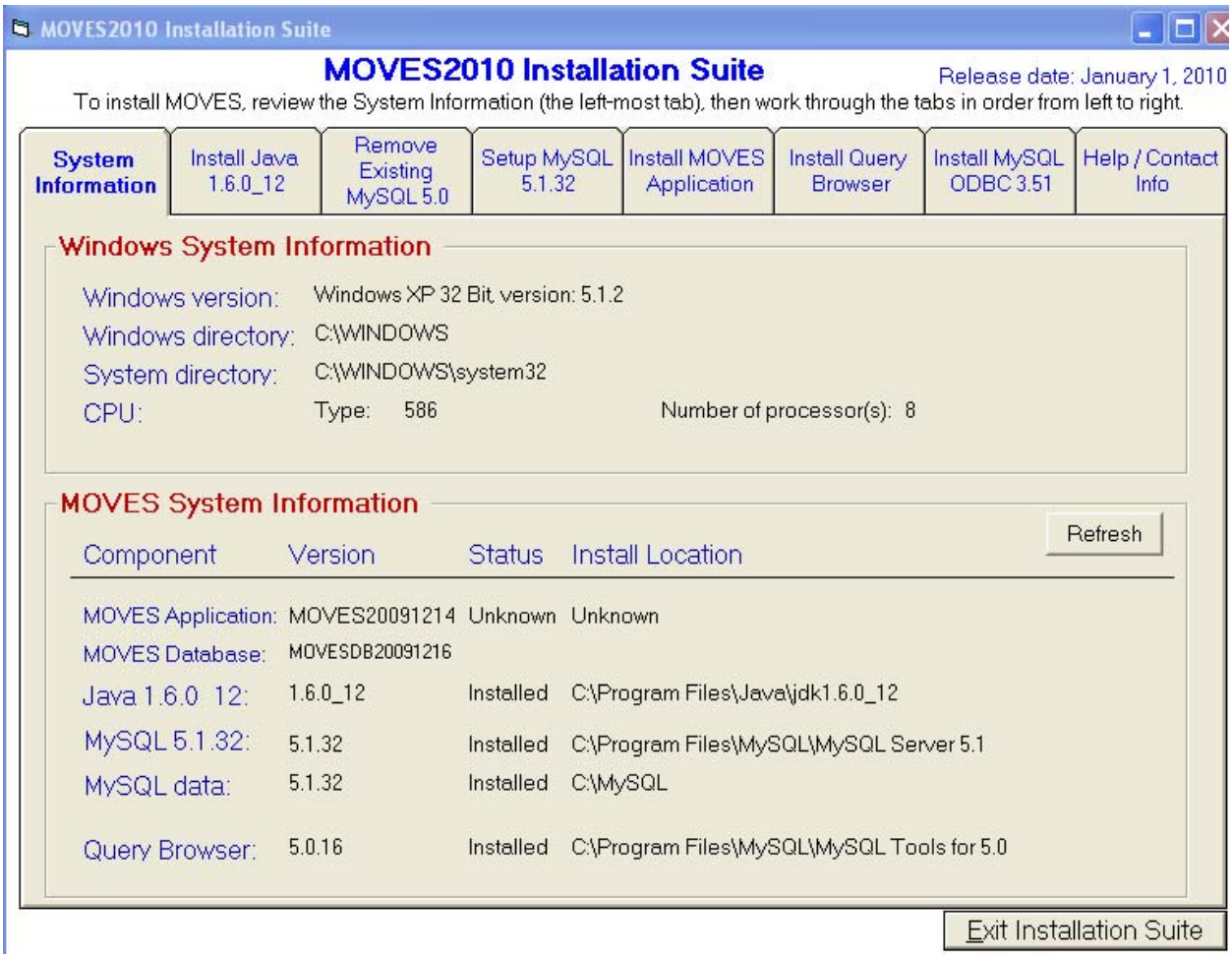
⇒ If you would like to work through an example, a set of sample scenarios is included in the Appendices.

⇒ If you would like to get started using MOVES for regular work, advice and guidance on using MOVES is available on the MOVES website.

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

1.3 Installation

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



1.3.1 System Requirements

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

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

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

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

1.3.2 Starting and Testing Your Installation

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

1. Check that the MySQL server is operating. This is done by opening the MS DOS prompt (Start-Programs-Accessories-Command Prompt) and changing the path to "C:\mysql\bin" (c:\> cd \mysql\bin). After changing the path, the user should type 'mysql' at the prompt. The MySQL program should begin. If an error message appears, the MySQL program or server has not been successfully installed. Once installed, the MySQL server will be set up to run automatically as a Windows system service.

2. Double-click the **MOVES Master** program icon.

3. When starting the **MOVES Master** program, the **About MOVES** panel will appear in the center of the GUI screen. Click the **"OK"** button to continue.

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

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

1.4 Executing Example Run Specification

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

1.4.1 Step-by-step Example Run

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

1. Select **File**, **Open** on the Main Menu Bar.

2. Navigate, if necessary, to the top level of the MOVES directory.

3. Select the **MOVES2010Example.mrs** file.

4. Select **Output**, then **General Output** on the Navigation Panel, which is the left-side panel. Select or click on Create Database to establish a database into which results should be placed. Select or enter a database name.

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

5. Select **Action**, **Execute** on the Main Menu Bar to start the RunSpec. The GUI will ask you if you want to save the RunSpec before executing. You may choose Yes, No or Cancel. The RunSpec will execute if either Yes or No are chosen, but selecting Cancel will not execute and will allow users to make changes to the RunSpec. This RunSpec may take up to 15 minutes to process.

6. The output will be saved in the MySQL relational database named in Step 4. The output can be reviewed using the Post-Processing menu options described in Section 2.3.5, below, or by viewing the database directly using MYSQL commands, either through the command prompt window or through MySQL Query Browser.

1.4.2 MOVESExecution Database

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

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

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

1.4.3 MOVES Output Structure

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

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

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

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

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

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

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

3. MovesRun: This table contains information about the date and time of the run, information about the run specification, and the name of the units in which MOVES outputs are represented. These include units for energy (i.e., joules), emissions (i.e., tons), VMT (i.e., miles), and time units (i.e., months, days, hours, etc.).

4. MovesError: This table contains any error messages or diagnostic information that might be generated if the MOVES run is unsuccessful.

5. MovesWorkersUsed: This table contains information as to which copy of theMOVES Worker Program processed portions of the run. It is of interest if the run is executed on a multi-computer configuration.

6. Activity Type: This table lists the activity types that can be reported in the movesactivityoutput table and provides their activitytypeid. In MOVES2010 this table lists: distance traveled, source hours, extended idle hours, source hours operating, source hours parked, population, and starts. (See also Section 2.2.10.1.3**)**

7. MovesEventLog: This table is used by MOVES programmers for storing diagnostic results. It contains no information useful to users.

8. RatePerDistance: When "Emission Rates" is selected on the Scale panel, the RatePerDistance table stores emissions as rates per distance (e.g. grams per mile) with the units depending on those selected in the run specification. This table includes rates for exhaust and evaporative emissions that occur while vehicles are on "real roads", but does not include rates for starts, extended idle, or evaporative emissions (with the exception of refueling) that occur while a vehicle is parked.

9. RatePerProfile: When "Emission Rates" is selected on the Scale panel, the RatePerProfile table stores vapor venting emissions from parked vehicles as rates per vehicle (e.g. grams per vehicle). Note that the denominator is the total vehicle population—not just the vehicles that are parked. The temperature profile id in this table refers to a particular daily pattern of temperatures, important because vapor venting depends on temperatures in both the current and previous hours.

10. RatePerVehicle: When "Emission Rates" is selected on the Scale panel, the RatePerProfile table stores emissions from starts and extended idle, and some evaporative emissions (permeation and liquid leaks) from parked vehicles as rates per vehicle (e.g. grams per vehicle). Note that the denominator is the total vehicle population—not just the vehicles that are parked.

11. **MovesTablesUsed:** This table contains a list of the tables used when executing the RunSpec and the source database for each table. The numbering of the tableUseSequence is such that lower numbers take priority. Some tables may be listed

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

2.0 Using MOVES

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

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

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

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

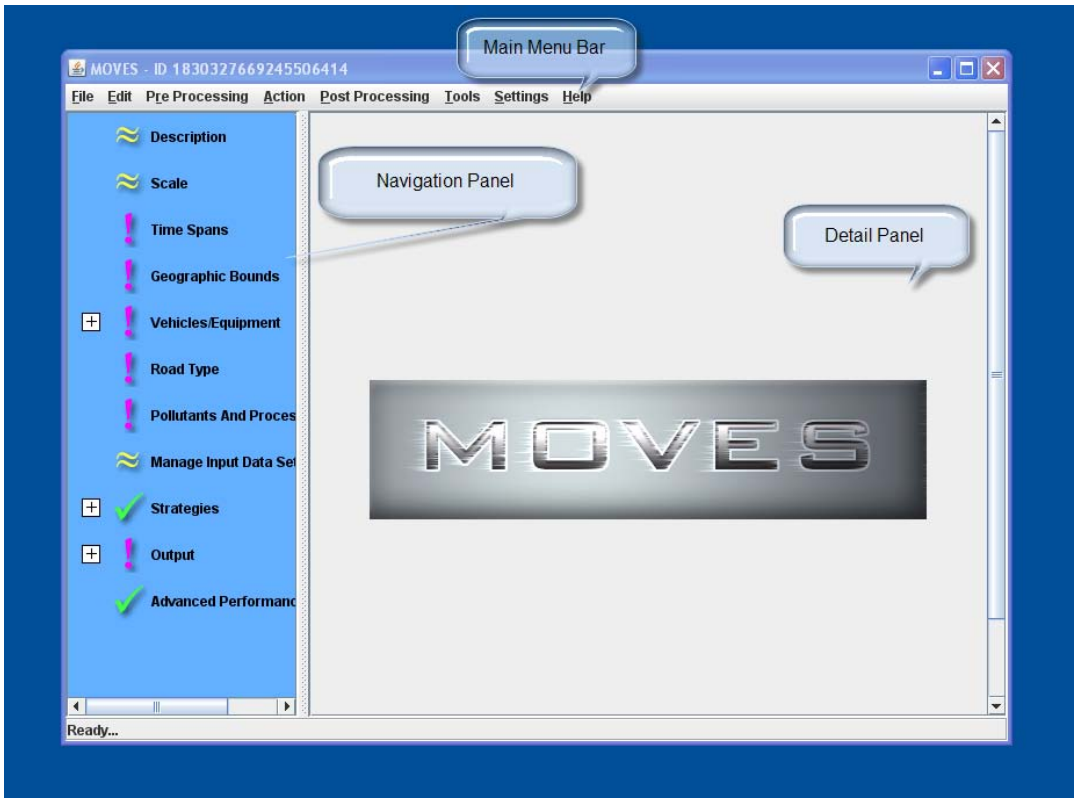
Several components of the model are intended to facilitate customization. These include the **Strategies** procedures described below. MOVES also allows users to provide **User Input Databases** and to use various **Importers** with the **County Data Manager**, and Project Domain Manager described later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

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

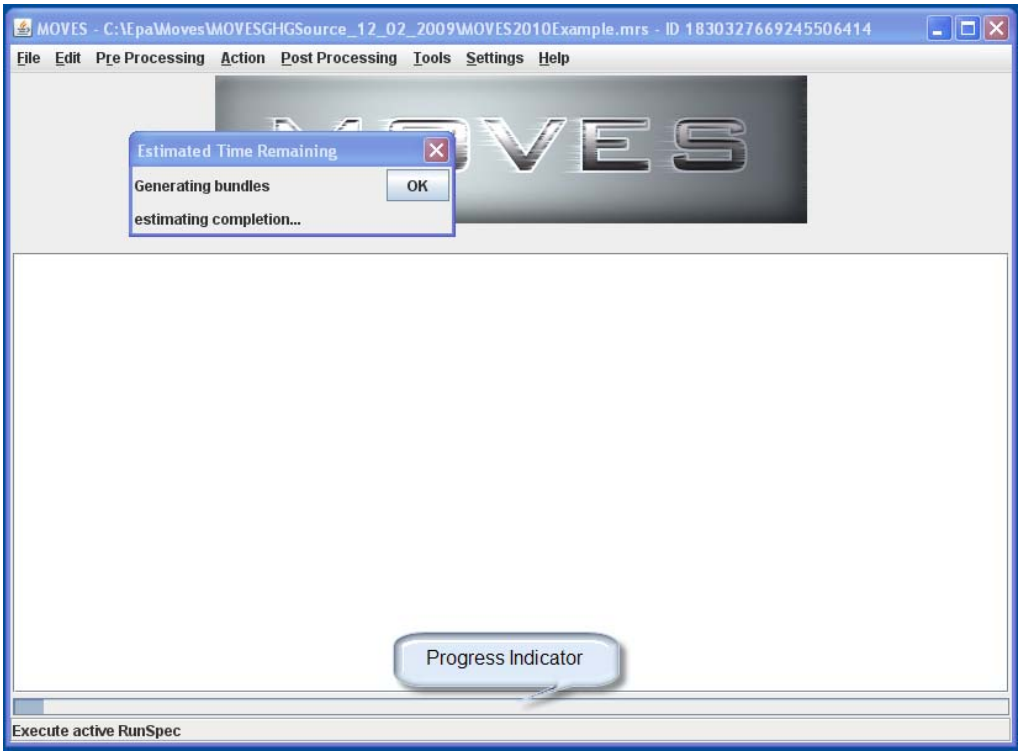
NOTE Many MOVES features request that the user create input or output databases. These database names must be consistent with MYSQL naming conventions. In general, this means they may contain letters, numbers, "_"(underscore) and "$". They may not contain spaces, special characters, or be "reserved words"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

2.1 Overview of MOVES User Interface

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



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



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

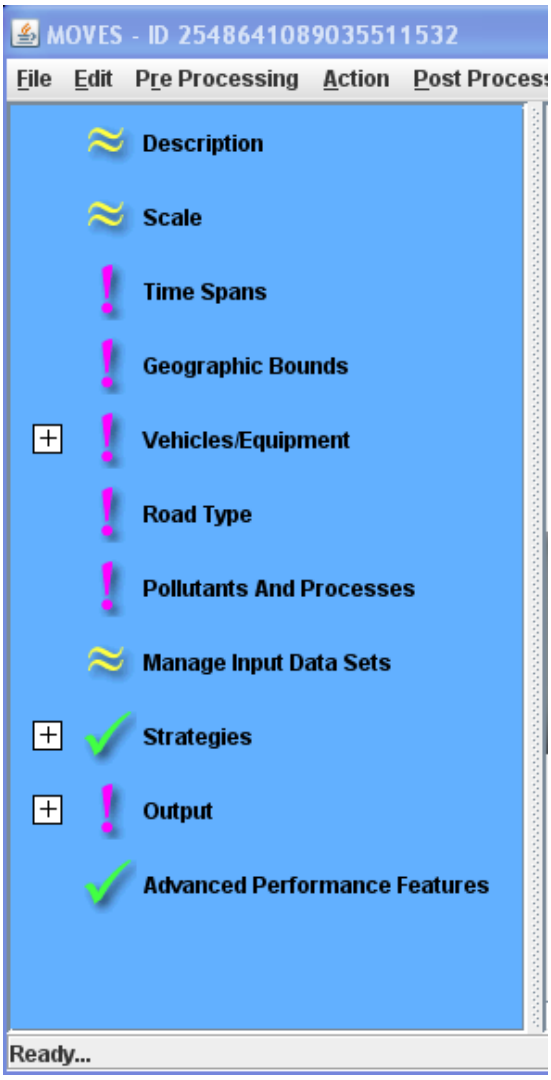
The MOVES2010 model features "mouse hover" user help throughout the model. To use it, the user simply places the mouse over a particular MOVES GUI icon or command and keeps it still. A small help box will appear with a brief text message; for example, the readiness status of a Navigation Panel icon,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

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

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

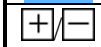
2.2 Navigation Panel

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



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

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

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

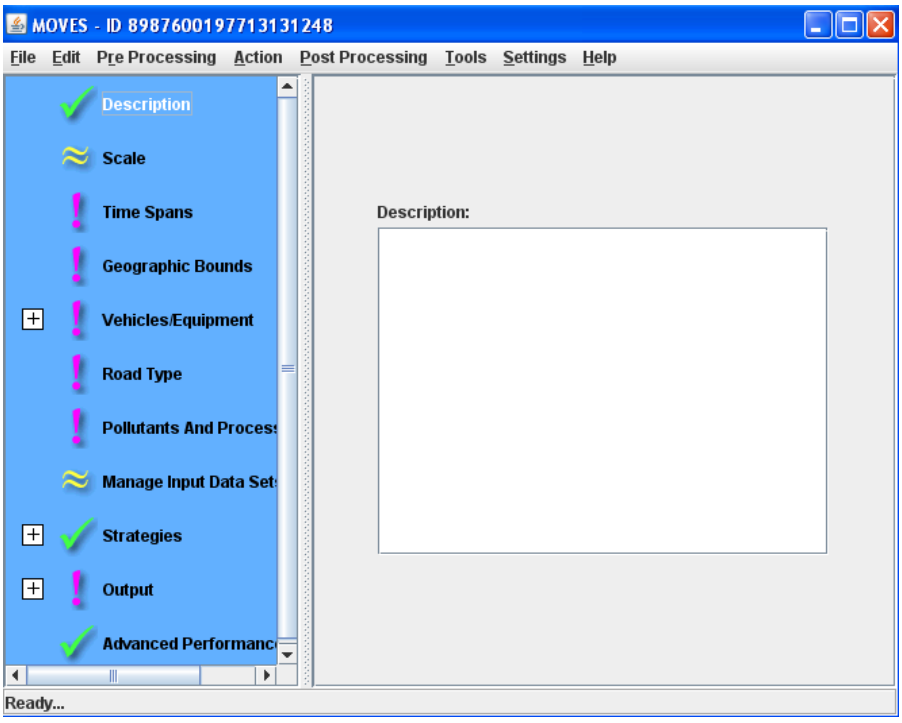
Note Parameters may be visited in any order, however some choices affect the choices available in other screens. In particular, changing the "Scale" parameters can impact other parameter choices. You may wish to save your Run Spec before switching Scales and to pay attention to the Navigation Panel icons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

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

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

2.2.1 Description

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



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

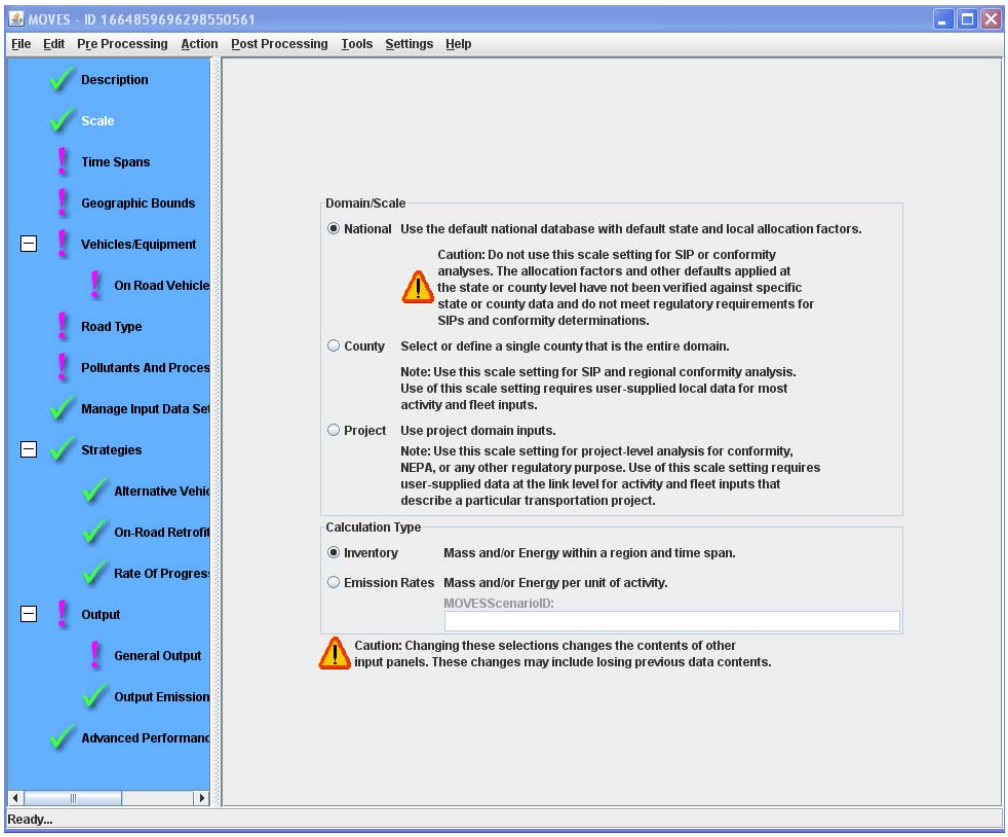
!Tip The default Description is blank and the MOVES model can be run without a Description. However, it is considered a best practice to carefully describe each and every version of a RunSpec. This will assist in documenting the purpose of that particular MOVES2010 run. See Appendix E, Best Practices,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

2.2.2 Scale

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

2.2.2.1 Domain/Scale

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



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

Project Domain/Scale is the finest level of modeling in MOVES. It allows the user to model the emission effects from a group of specific roadway links and/or a single off-network common area. The definition of a roadway link is a section of any road where a vehicle is moving for more than three seconds. An off-network common area may include project boundaries where vehicle starts and extended idling emissions are produced. The use of the Project Domain/Scale requires the user to completely define the individual project (i.e., specify all individual roadway links and/or the off-network common area). All of the project level data is entered in the Project Domain Manager (see Section **2.3.3.2)**for data input details). If specified correctly, the MOVES Project Domain/Scale will utilize MOVES emission rates and other factors to correctly calculate emission inventory results for the user's defined project.

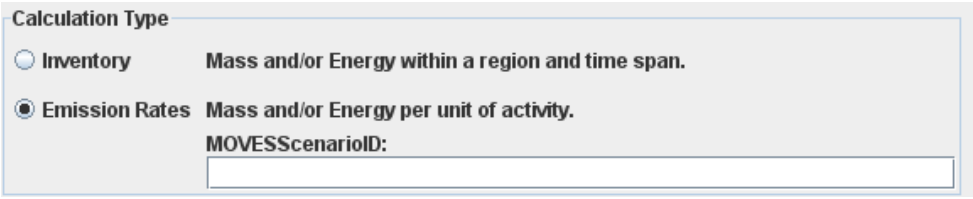
Note MOVES2010 is not capable of modeling any evaporative emission processes when using **Project Domain/Scale**. EPA plans to add this feature in future versions of MOVES.

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

2.2.2.2 Calculation Type

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

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



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

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

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

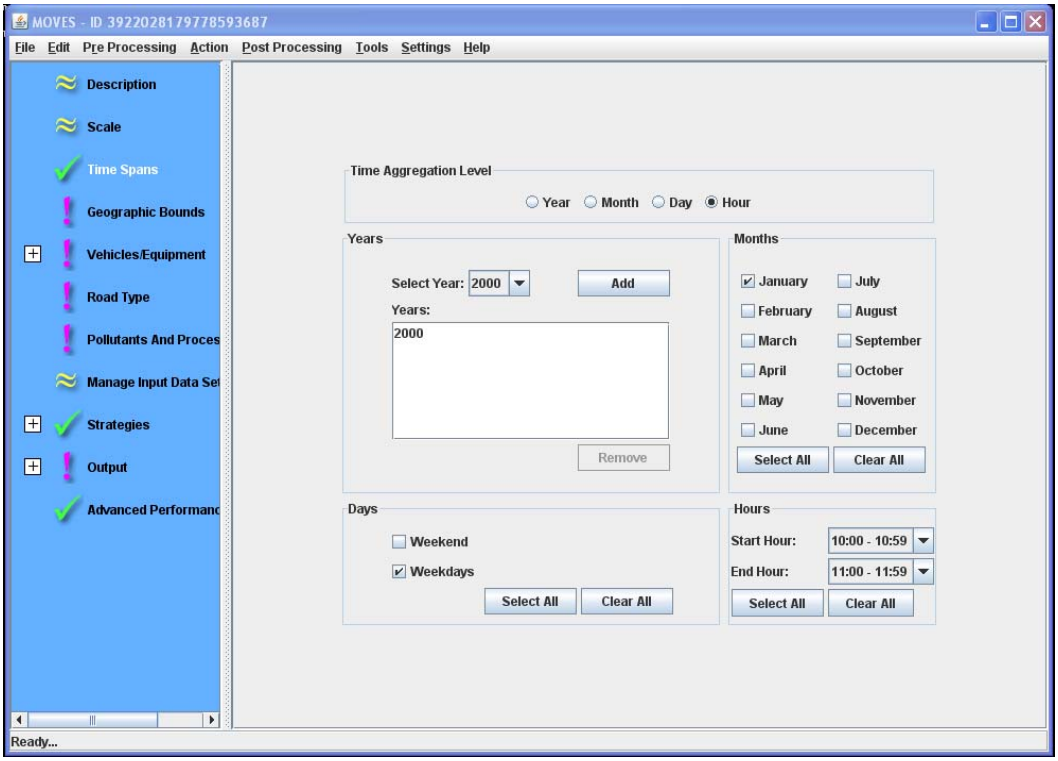
1) If **National Domain/Scale** is selected, only **"Zone & Link**" can be chosen in the "**Region**" section of the **Geographic Bounds** panel, or 1b) if **County or Project Domain/Scale** is selected, only "**Zone & Link**" or "**Custom Domain**" can be chosen;

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

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

2.2.3 Time Spans

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



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

2.2.3.1 Time Aggregation Level

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

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

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

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

2.2.3.2 Years

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

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

2.2.3.3 Months

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

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

2.2.3.4 Days

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

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

2.2.3.5 Hours

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

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

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

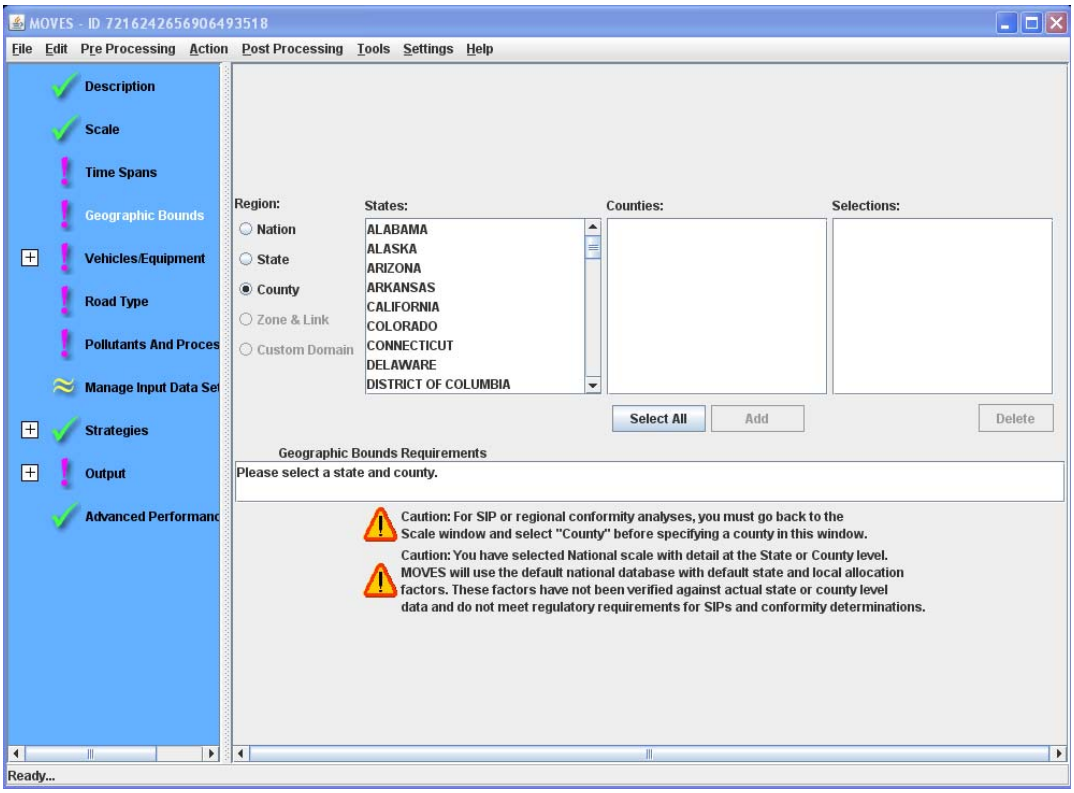
Note If the **Project Domain/Scale** is selected, the Run Spec may cover only a single hour..

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

2.2.4 Geographic Bounds

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

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



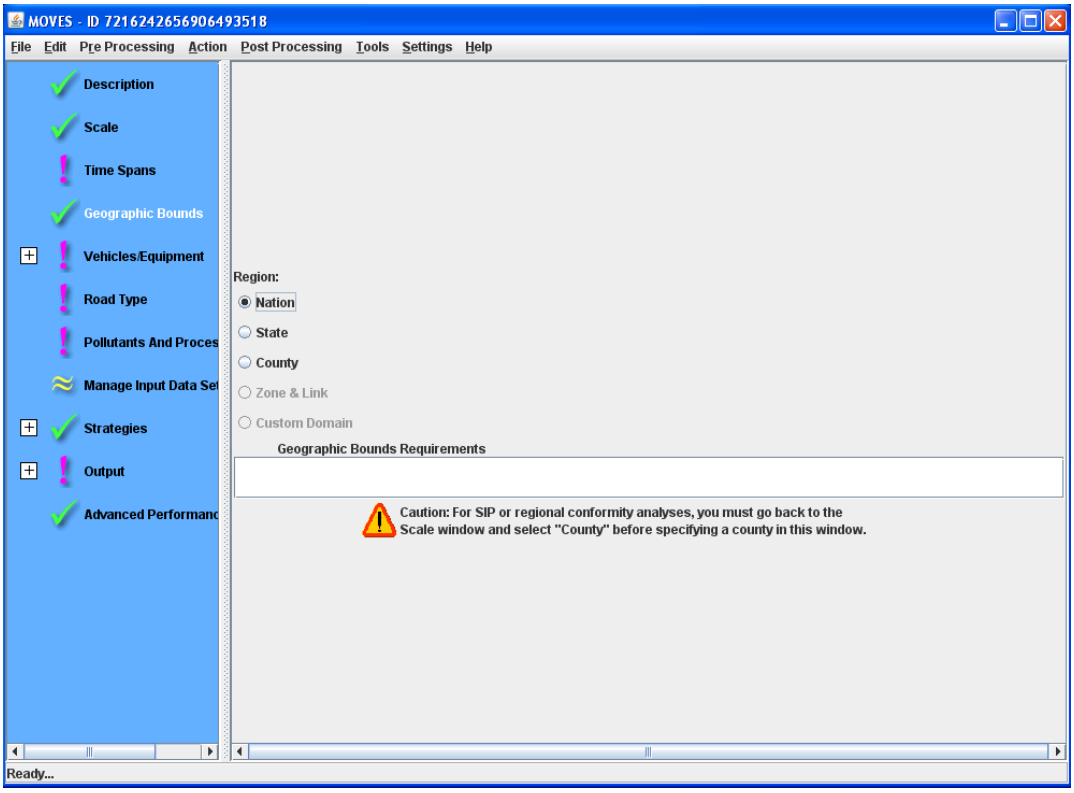
Three buttons, "**Nation**", "**State**" and "**County**" are normally available. Choose the one appropriate for your desired run.

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

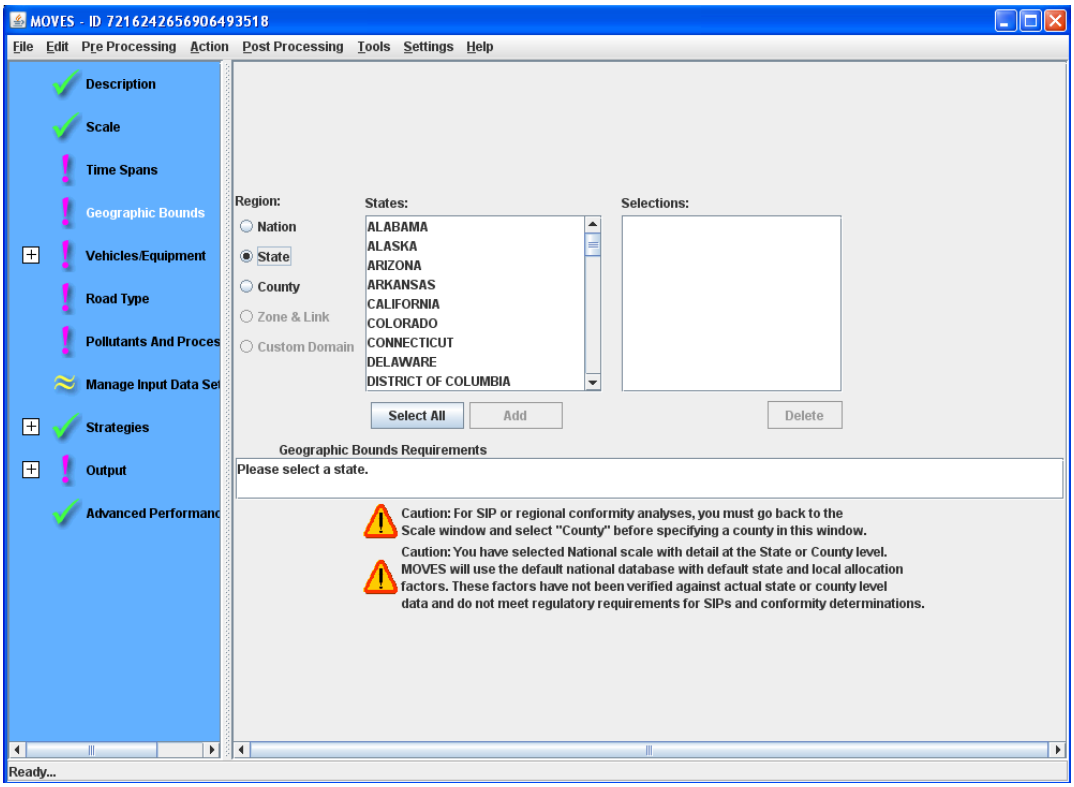
For example, if the user chooses **"State"** and **"Michigan,"** the model will average the temperatures of all Michigan counties into a single average set (it will be a set because the temperatures will not necessarily be averaged by time) and perform the simulation with these average values. The report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

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

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



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



To choose the desired state(s):

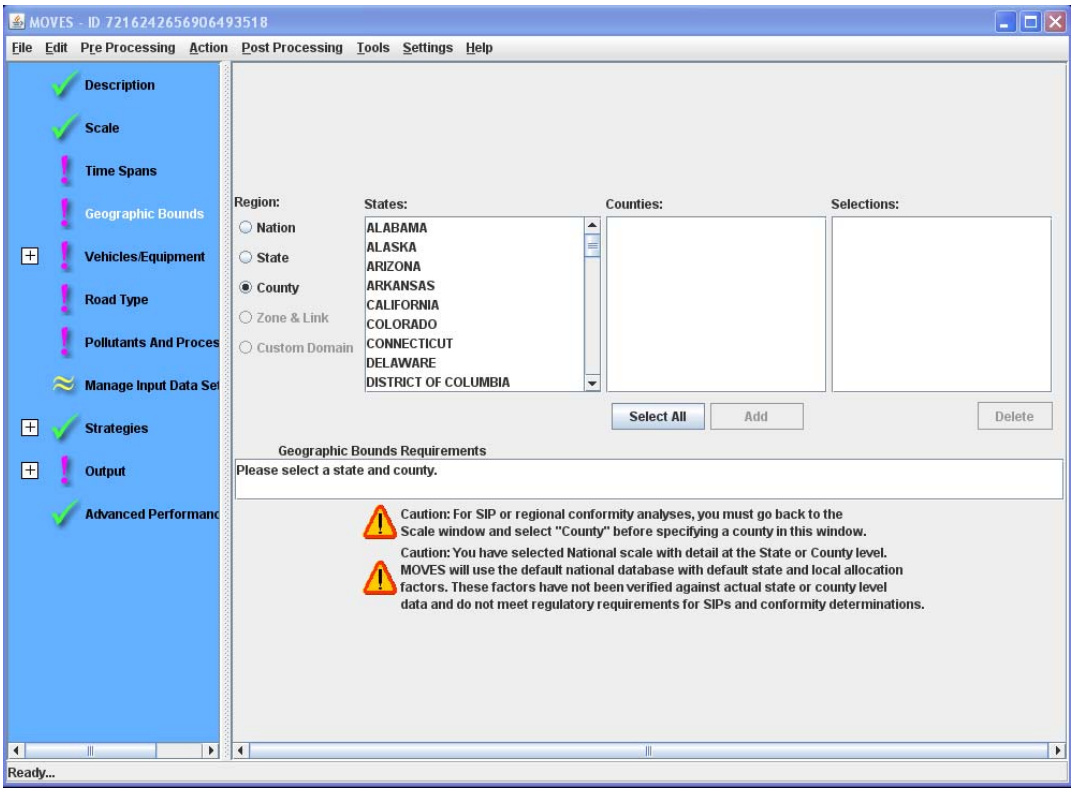
1. Scroll to desired state in **"States"** box and click on it to highlight. Click the **"Add"** button under box. State will appear in the "Selections" box.

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

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

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

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



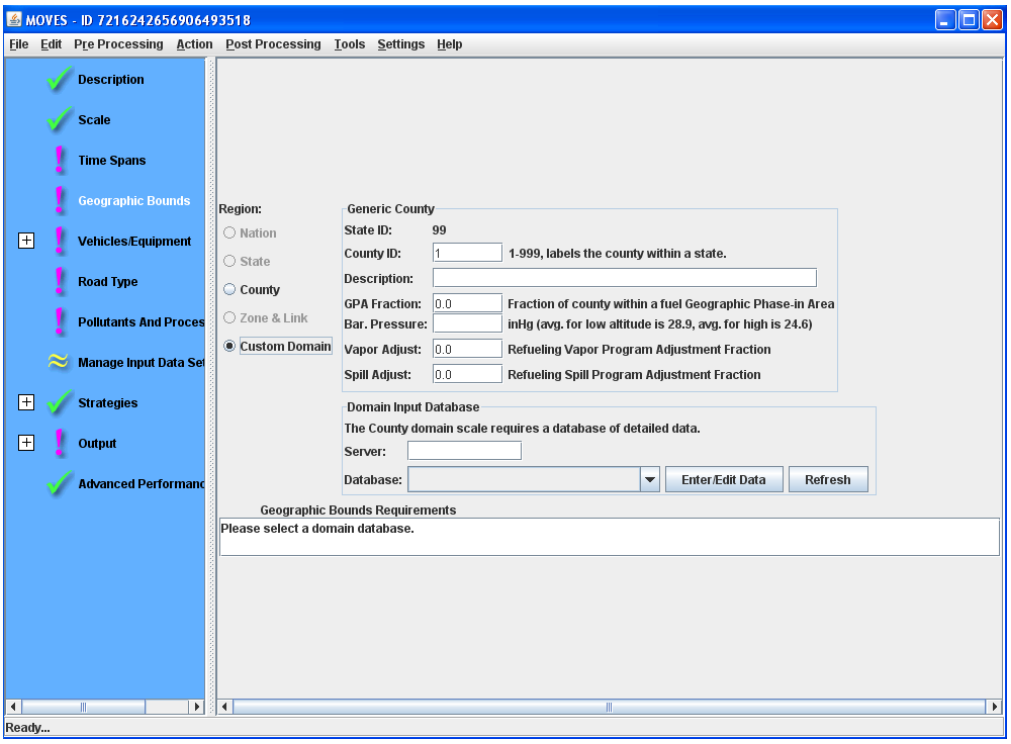
The process for selecting a specific county is similar to selecting a state, outlined above.

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

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

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

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

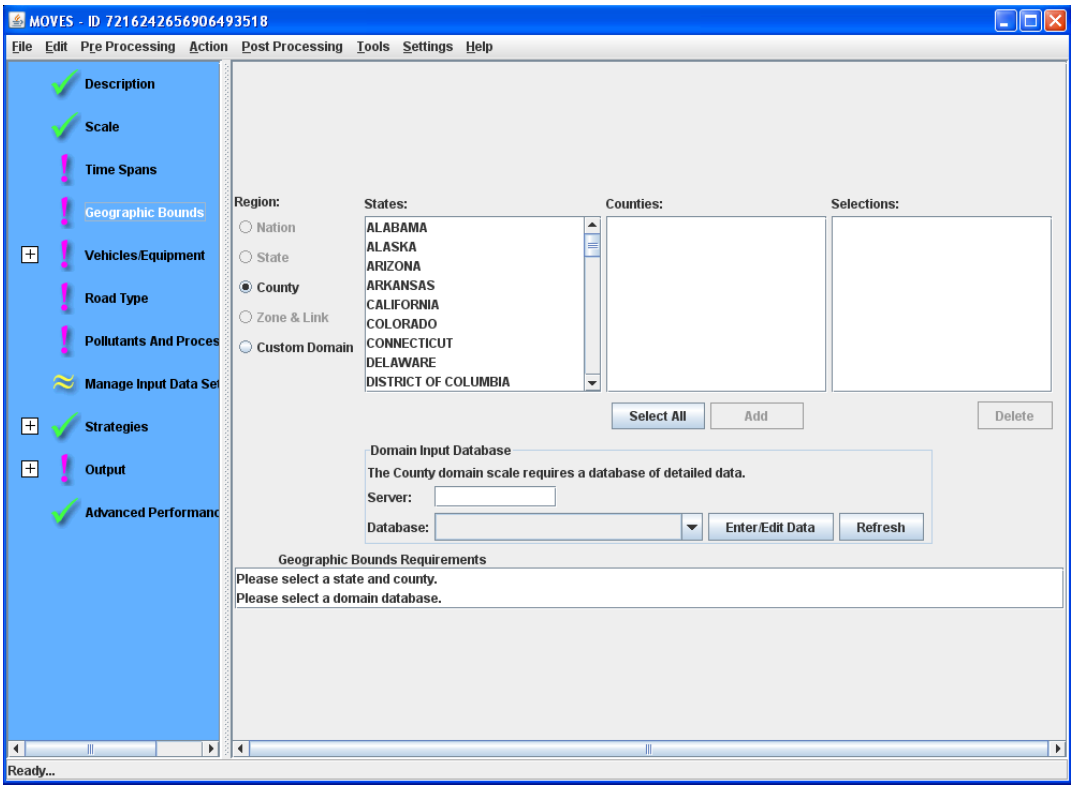


In this case, the user must provide a short county identifier (1-999) that will combine with the state ID (99 for generic) to form the standard 5 digit county identifier used throughout MOVES. In addition, the user must provide a description and GPA (Geographic Phase-in Area) fraction. The Geographic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

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

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

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



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

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

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

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

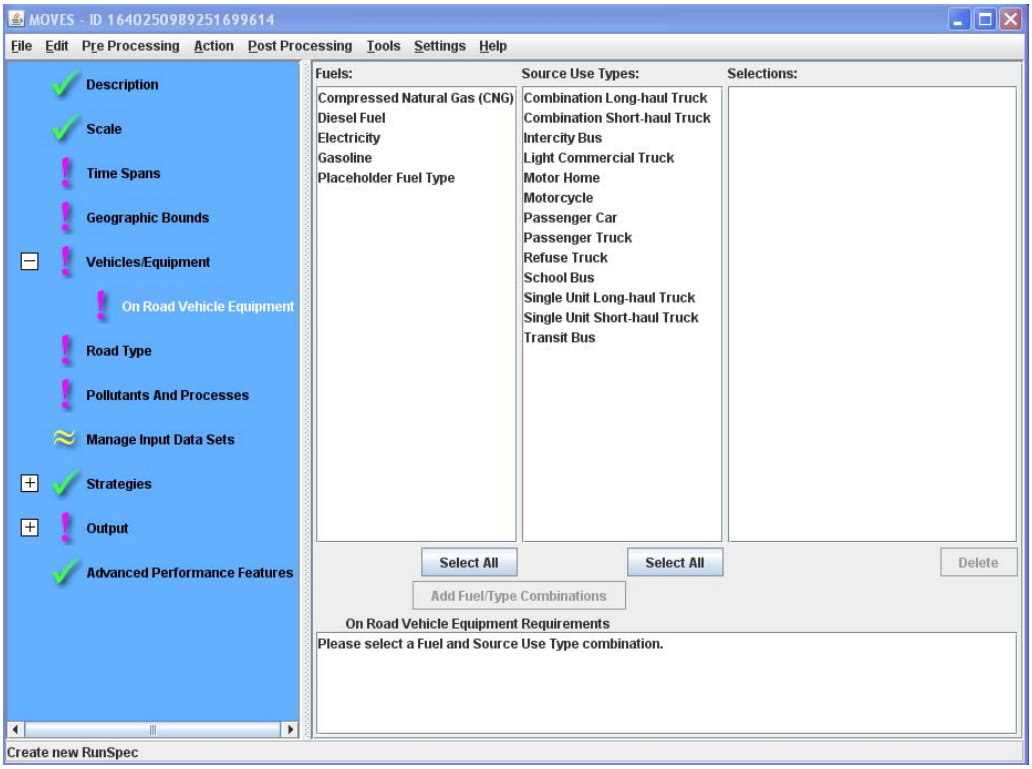
The user can clear the selected database and update the list of available databases by clicking the "**Refresh**" button.

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

2.2.5 Vehicles/Equipment

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

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



1. Click and highlight the **Fuels** choice or click the "**Select All**" button to choose all the choices.

2. Click and highlight the **Source Use Types** choice or click the "**Select All**" button to choose all the choices.

3. Click "**Add Fuel/Type Combinations**" button to move the selected choices to the **Selections** box.

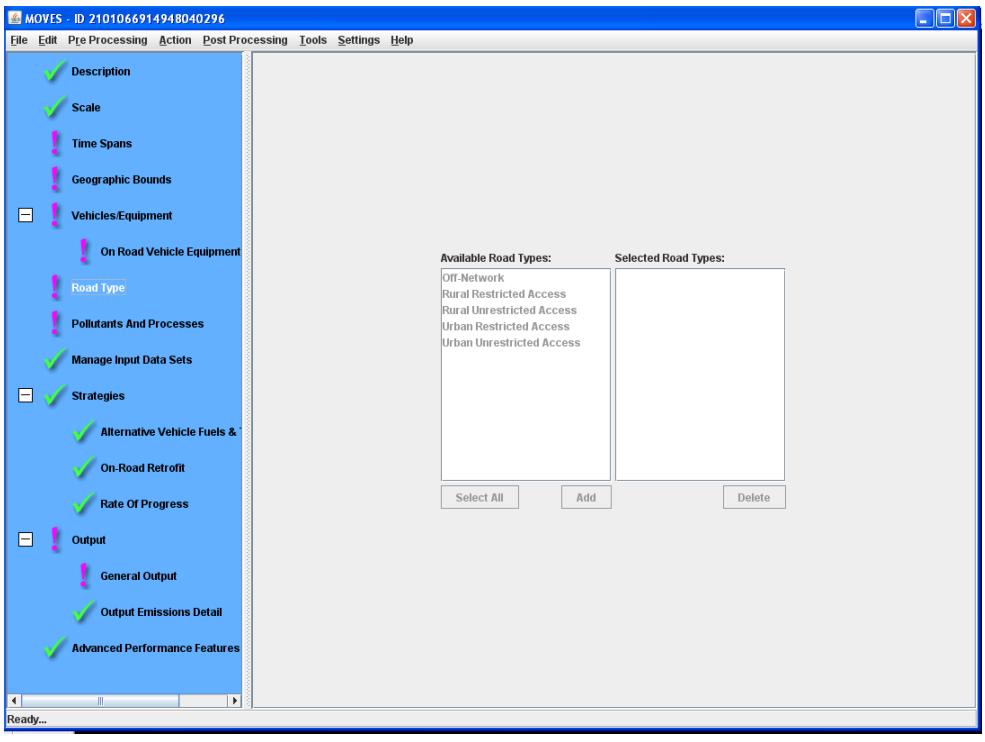
To clear highlighted selections from Selections box click **"Delete"**

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

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

2.2.6 Road Type

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



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

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

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

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

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

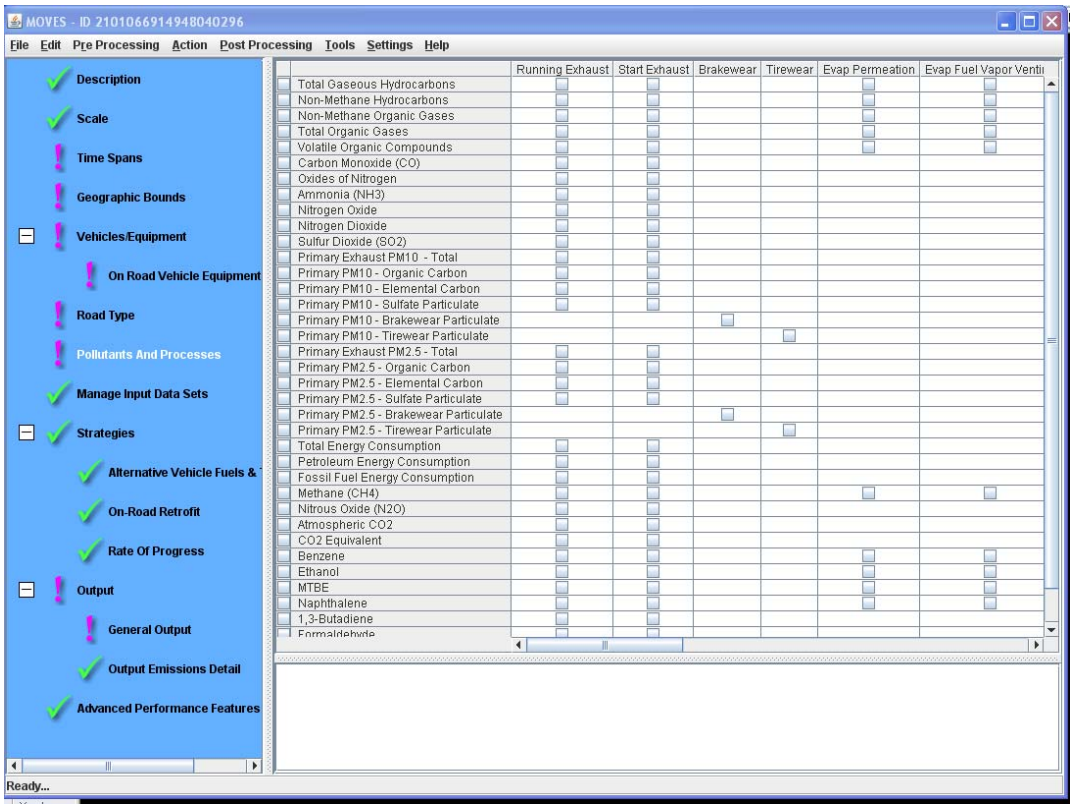
Note "**Off Network**" is automatically selected when start or extended idle processes are selected on the "**Pollutants and Processes**" panel. All **Road Types** are automatically selected when any evaporative or refueling processes are selected on the "**Pollutants and Processes**" panel.

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

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

2.2.7 Pollutants and Processes

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



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

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

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

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

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

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

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

2.2.7.1 Processes

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

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

2.2.7.2 Pollutants

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

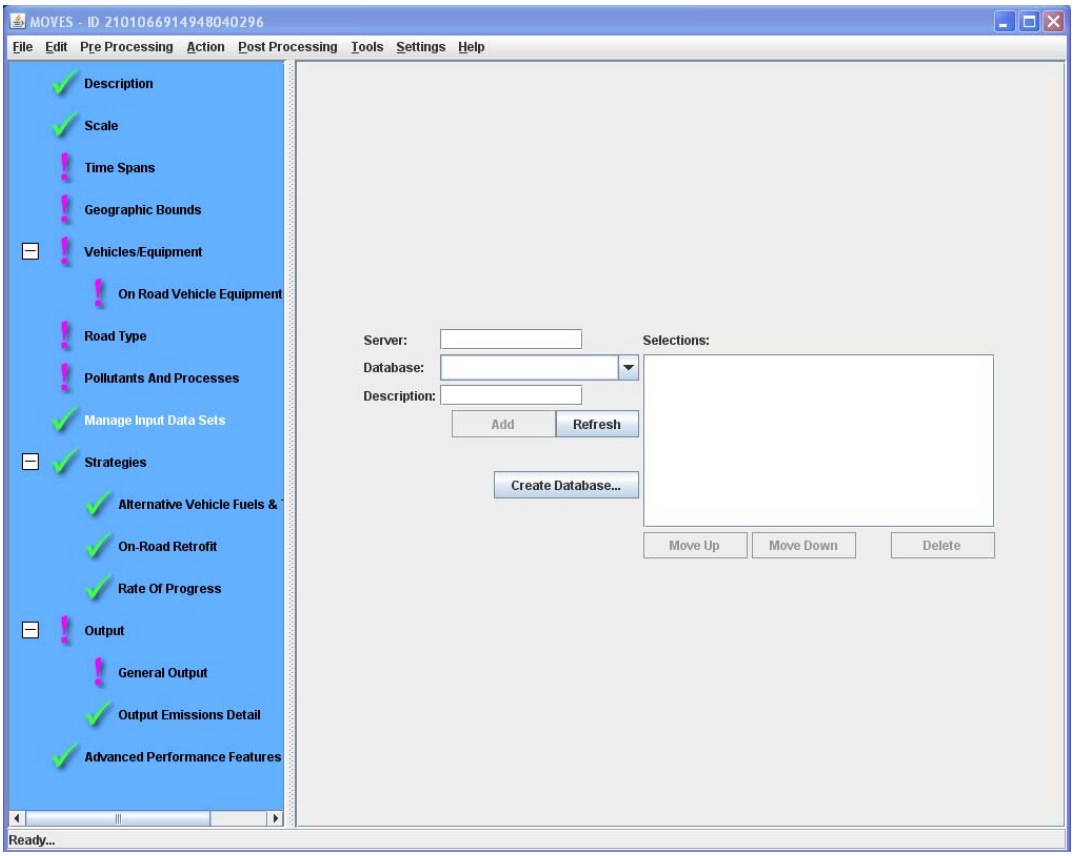
Pollutant Name	Abbreviation	Includes	Depends on
Total Gaseous Hydrocarbons	THC		
Non-methane Hydrocarbons	NMHC		THC and CH_4
Non-methane Organic Gases	NMOG		NMHC
Total Organic Gases	TOG		NMOG and CH_4
Volatile Organic Compounds	VOC		NMHC
Carbon Monoxide	CO		
Oxides of Nitrogen	NO_x	NO,NO_2	
Ammonia	NH_3		
Nitrogen Oxide	NO		NO_x
Nitrogen Dioxide	NO_2		NO_x
Sulfur Dioxide	SO_2		
Primary PM10 Total	PM10	PM10 OC PM10 EC PM10 SO_4	
PEPM10 Organic Carbon	PM10 OC		PM2.5 OC
PEPM10 Elemental Carbon	PM10 EC		PM 2.5 EC
PEPM10 Sulfate Particulate	PM10 SO_4		Total Energy Consumption
PEPM10 Brakewear Particulate			
PEPM10 Tirewear Particulate			
Primary PM2.5 Total		PM2.5 OC PM2.5 EC PEPM2.5 SO_4	
PEPM2.5 Organic Carbon	PM2.5 OC		
PEPM2.5 Elemental Carbon	PM2.5 EC		
PEPM2.5 Sulfate Particulate	PM 2.5 SO_4		Total Energy Consumption
PEPM2.5 Brakewear Particulate			
PEPM2.5 Tirewear Particulate			
Total Energy Consumption	TEC		
Petroleum Energy Consumption	PEC		Total Energy Consumption
Fossil Fuel Energy Consumption	FFEC		Total Energy Consumption
Methane (CH_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	C_6H_6		VOC
Ethanol	C_2H_6O		VOC
Methyl tertiary butyl ether (MBTE)	$C_5H_{12}O$		VOC
Naphthalene	$C_{10}H_8$		PM10 Total THC
1,3 Butadiene	C_4H_6		VOC
Formaldehyde	CH_2O		VOC
Acetaldehyde	C_2H_4O		VOC
Acrolein	C_3H_4O		VOC

Table 2.2.7.2

2.2.8 Manage Input Data Sets

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

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



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

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

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

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

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

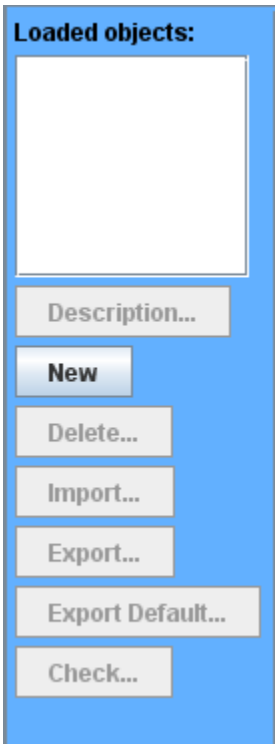
!Tip Databases created with the **County Data Manager** or **Project Domain Manager are selected in the Domain Input Database** pane of the **Geographic Bounds** panel rather than the **Manage Input Data Sets** panel.

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

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

2.2.9 Creating and Using Internal Control Strategies

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



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

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

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

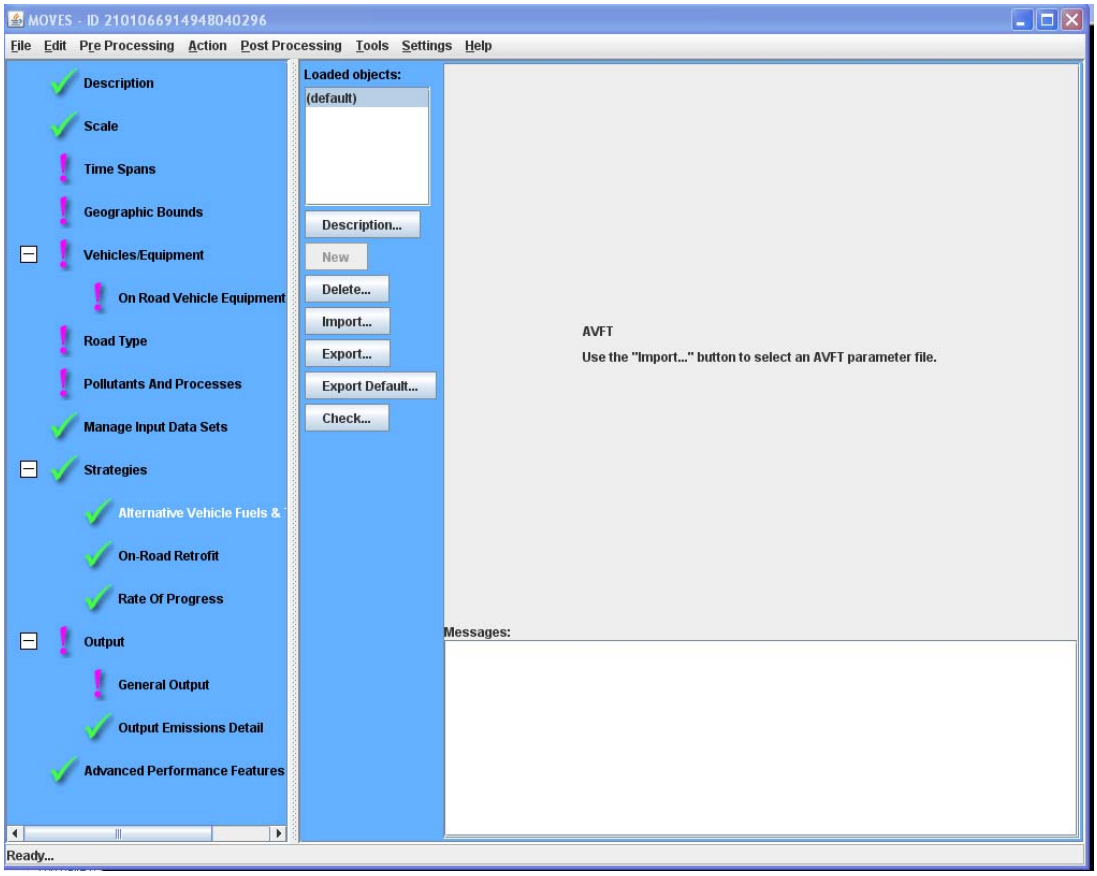
2.2.9.1 Adding Strategy Files

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

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

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

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 Default**." To create a template file for the On-Road Retrofit strategies, the user chooses "**New,**" and then exports a file containing only headers. Both the AVFT and the On-Road Retrofit Strategies allow the user to export previously imported files.

2.2.9.4 Naming Strategy Files

By clicking the "**Description**" button, the dialog box shown below is displayed:

Changing the description will change the text shown in the "**Loaded Objects**" window. However, importing different files will not change the description.

!Tip The **Desciption** text can be misleading since it must be manually updated if new strategy files are loaded. Users may wish to leave it blank.

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. However, the version of the file embedded within the RunSpec is set at the time of the import; in other words, if any change are made to the strategy file outside of MOVES, the file must again be imported for the RunSpec to reflect the changes made.

2.2.9.6 Creating 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 row contains field names: "SourceTypeID", "ModelYear", "fueltypeID", "EngTechID" and "FuelEngFraction". Default SourceTypeIDs and FuelTypeIDs are defined in the SourceUseType and FuelType tables of the MOVES Default Database and summarized in **Appendix J** of this user guide. "ModelYear" refers to the model year of the vehicles. "EngTechID" refers to the engine technology type, described below. "FuelEngFraction" 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	Available in MOVES2010?
1	Conventional Internal Combustion	Yes
2	Advanced Internal Combustion (Improved fuel economy)	Yes
11	Moderate Hybrid - Conventional Internal Combustion	No
12	Full Hybrid - Conventional Internal Combustion	No
20	Hybrid - Advanced Internal Combustion	No
21	Moderate Hybrid - Advanced Internal Combustion	No
22	Full Hybrid - Advanced Internal Combustion	No
30	Electric	Yes, required with "Electricity" fueltype
40	Fuel Cell	No
50	Hybrid - Fuel Cell	No

Table 2.2.9.6

Beneath the field names are data rows. For MOVES2010, there are about 2230 rows in an AVFT file if all source types are selected. Users can add rows for new fuel/engine technology combinations if they are allowed for that sourcetype, or delete rows for combinations that the user does not want included. Once the user has decided upon a complete set of fuel and engine technologies for a source type/model year combination, the user should edit the "FuelEngFraction" in each row so that the fractions entered sum to 1 within the source type/model year combination.

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.

If all years do not sum to one, the panel will display an error message and an option to "**Normalize**" the data. Normalizing will proportionally adjust the entries so that each sourcetype/modelyear combination sums to one.

!Tip If you "Normalize" imported AVFT data, it can be useful to export and view the normalized data to make sure that the normalization adjustments lead to the vehicle mix you intended

2.2.9.7 Managing On-Road Vehicle Retrofit Data

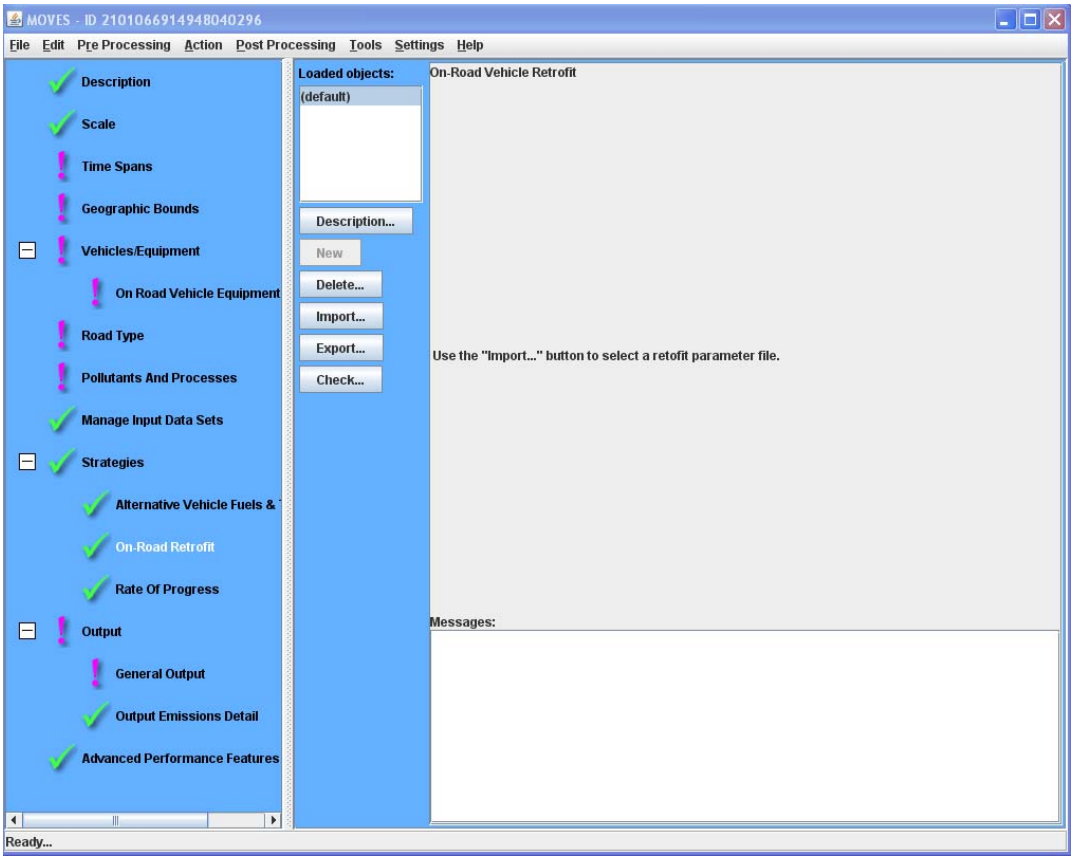
MOVES2010 has the capability to model on-road vehicle retrofit strategies for all exhaust pollutants for diesel trucks and buses. **Appendix D** provides detailed instructions for creating an **On-Road Retrofit** data file.

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

the **Strategies** section. The "**Description**", "**New**", "**Export**", "**Delete**", and "**Check**" buttons work as described earlier.

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 is similar to the AVFT import and is described below.



2.2.9.8 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

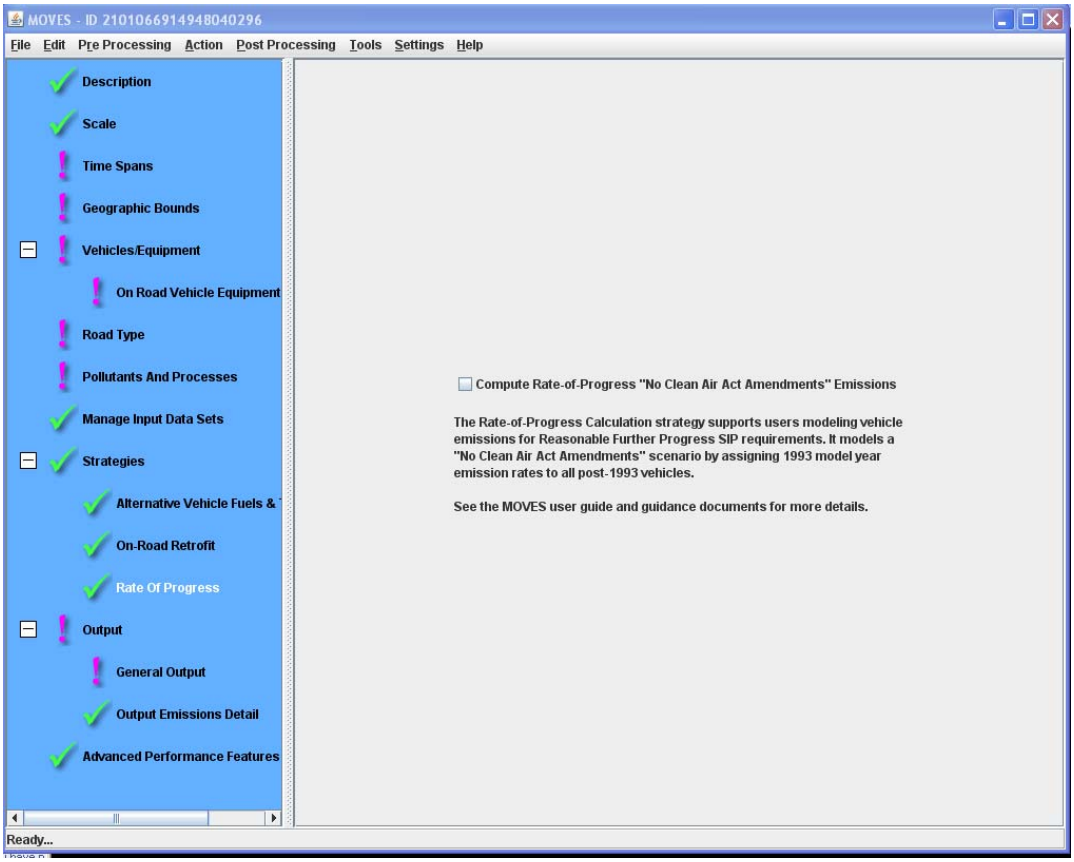
Once data are imported, the display in the Detail Panel will show the name and location of the imported file ("Data Source"). A "**Reload**" button that will also appear next to the file name can be clicked on to capture any changes to the imported data file.

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 must check, the **Use the On-Road Retrofit Settings** 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.

CAUTION! If the user does not select **Use the On-Road Retrofit Settings**, the retrofit inputs will not be used.

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. Importing data for retrofits requires that the user enter the description of the field (e.g. NOX rather than pollutant 3).

2.2.9.9 Rate of Progress Calculation



If the user selects "Compute Rate-of-Progress 'No Clean Air Act Amendments' Emissions, the MOVES model will assign 1993 model year emission rates to all post-1993 vehicles.

Under the Clean Air Act, some State Implementation Plans (SIPs) must include a demonstration that the state is making "reasonable further progress" in reducing hydrocarbon and

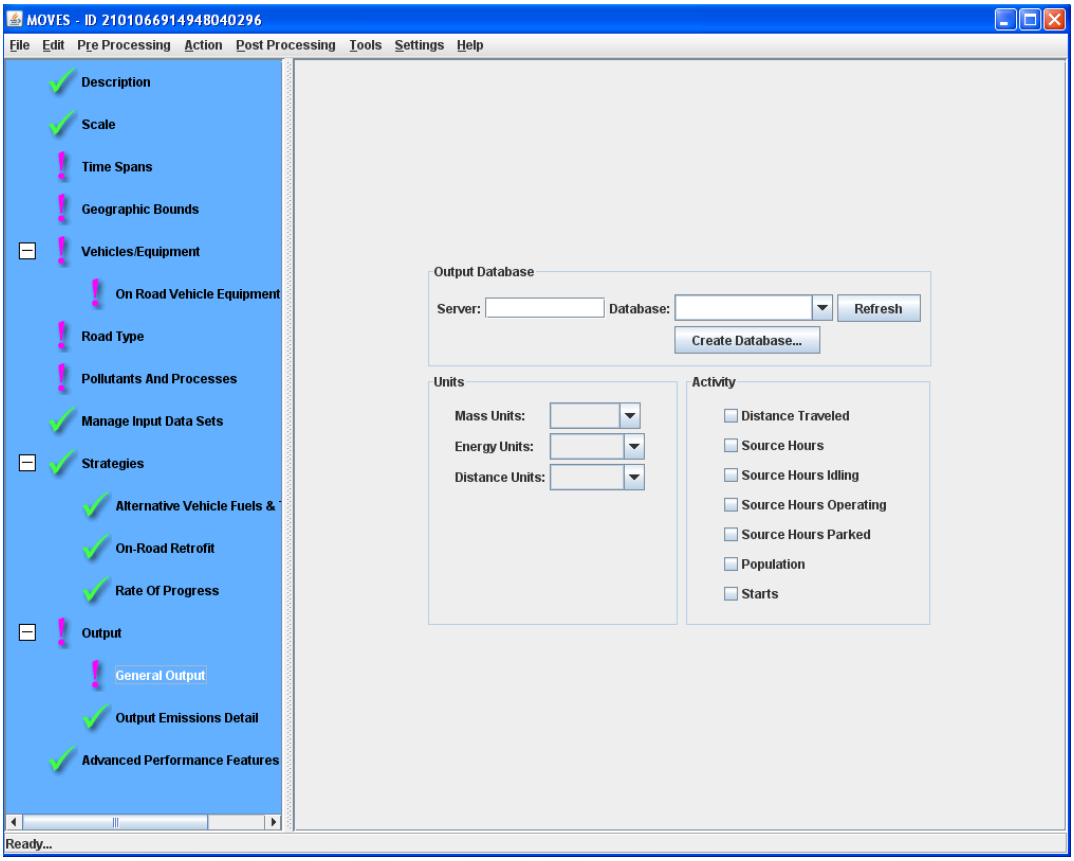
NOx emissions. To compute the rate of progress to this goal, the Act requires calculating emissions with and without the impact of the Clean Air Act Amendments of 1990. To perform the highway vehicle portion of these calculations with MOVES, users should use the Rate of Progress feature. They should also set inspection maintenance programs and fuel programs to appropriate parameters. For more information, see the MOVES Technical Guidance.

2.2.10 Output

Selecting **Output** on the Navigation Panel expands to show **General Output** and **Output Emissions Detail**. Each of these panels 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 units, and the activity types reported. The panel has three sub-panels: **Output Database, Units**, and **Activity**.



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 data folder 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 never attempt to create a new output database with the same name as the MOVES default database or of any user-supplied MOVES input databases.

!Tip The output database should follow a naming convention to differentiate it from other databases; typically, using "_out" at the end of the database name is recommended.

2.2.10.1.2 Units

The **Units** box contains user choices for the mass, energy, and distance units in which the results will be reported. The choices for mass are kilograms, grams, pounds, or US tons (short tons). The available energy units are Joules, Kilojoules, or Million BTUs (British Thermal Units). The available distance units are miles or kilometers. The user may choose only one unit for each of the three measurement 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 narrow spatially, temporally, and in terms of vehicle and road types, then choosing "**Tons**" as the "**Mass Units**" may not give you the desired level of detail.

2.2.10.1.3 Activity

The Activity panel allows users to choose which vehicle activity to report in the MOVESActivityOutput table.

If **"Emission Rates"** is selected on the **"Scale"** panel, **"Distance Traveled"** and **"Population"** will be selected automatically since their values are needed for the rates calculations.

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

NOTE In the MOVESActivityOutput table, population always has month, day & hour keys of zero (i.e. "doesn't matter") because population is assumed constant in time across an entire year. This scheme reduces table size, but requires care when filtering results and

when joining to other tables. Also, if you assign some VMT to a road type, but then do not select that road type in the RunSpec, the population will be lower than it should be. This is because the missing VMT is not used when calculating population. Users should model all road types (where VMT is allocated) to ensure proper calculations.

NOTE Activity is generated only when at least one process that uses that activity is chosen on the Pollutants/Processes panel. If the user chooses an activity but does not choose the associated process, the MOVESActivityOutput table will not include the desired results.

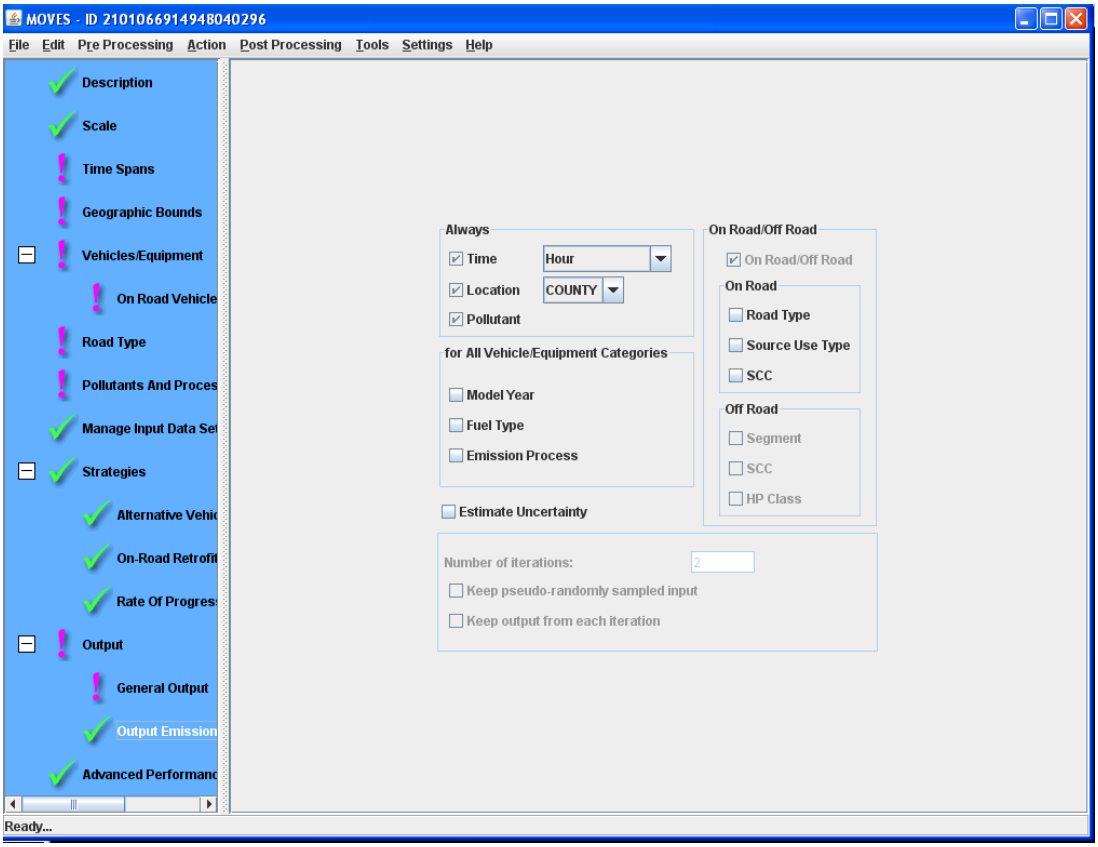
The following table 2.2.10.1.3 lists and describes the activities MOVES can output, their units, and the processes that generate their information.

ActivityTypeID	Description	Units	ProcesseID	Notes
1	Distance traveled	Distance units as set in a RunSpec	1	Also needed for proper calculation of evaporative emissions when doing "Rates" calculations
2	Source Hours (population times the number of hours)	Hours, regardless of time unit aggregation	11, 12, 13	
3	Extended Idle Hours (the number of hours that combination long haul trucks spent in extended idle mode)	Hours, regardless of time unit aggregation	90	Offnetwork roads only
4	Source Hours Operating (the number of hours that vehicles spent operating)	Hours, regardless of time unit aggregation	1, 9, 10, 11, 12, 13	
5	Source Hours Parked (the number of hours that vehicles were parked)	Hours, regardless of time unit aggregation	11, 12, 13	Offnetwork roads only
6	Population (the number of vehicles—assumed constant for each year)	SourceUseType count	1, 2, 9, 10, 11, 12, 13, 90	
7	Starts (the number of engine starts)	Number of starts	2	Offnetwork roads only

Table 2.2.10.1.3

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**, **On Road/Off Road**, and **Estimate Uncertainty**.



Note Selecting detailed output here and using MYSQL post-processing to aggregate the output to the desired level allows the user control over the aggregation level without re-running the model. This is often the best choice for intial investigation of a question and smaller runs. However, if a run includes many times, locations, source types, etc., the user may want to reduce the number of items selected on this screen to avoid producing unmanageably large output files and longer post-processing queries.

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 "**Time**" 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). Similarly, the "Hour" selection will result in hourly output for each kind of day requested in the RunSpec. 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.

Note The behavior of the aggregation function of the **Time** selection may cause unexpected results. For example, if only two hours of the day are selected in the **Time Spans** panel, but "**24-Hour Day**" is selected in the **Output** panel, the daily results will only be the sum of the two selected hours, not all 24. Similarly, selecting only weekdays in the **Time Spans** panel, but selecting "**Month**" in the **Output** panel will result in a monthly total that only includes weekdays. See the MOVES Software Design Reference Manual for more information about how MOVES aggregates output.

By default, the "**Location**" button will report the same geographic aggregation level as selected in the **Geographic Bounds** panel. The user may change the aggregation 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 MOVES pollutants are always reported separately in the output. Where aggregation of pollutants is available, the aggregate is listed as a separate pollutant on the **Pollutants And Processes** panel. (For example, "**Primary Exhaust PM2.5 – Total**.")

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

MOVES2010 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.

Caution! If "**Custom Domain**" is chosen on the **Geographic Bounds** panel and output is distinguished by "**SCC**", the user must manually populate the "sccRoadTypeDistribution" table in MySQL and input it as a user-input database in the **Manage Input Data Sets** panel. As stated earlier, the default database contains no default data about custom domains, but this table cannot be edited in the **County Data Manager** so it must be provided separately by the user. Additionally, while there is default information for individual counties, users may want to review the "sccRoadTypeDistribution" table to determine how the MOVES road types translate to the SCC roads types for the county of interest.

!Tip If "**Emission Rates**" are chosen on the **Scale** panel, output should be differentiated by "**Source Use Type**". Doing so allows VMT, Road Type Distribution, and Average Speed Distribution to become placeholders (i.e., they must still be imported, but their values do not impact the results); however, if output is not differentiated by source type, the emission rate becomes a weighted average of the source types selected in the RunSpec and the three inputs mentioned will impact the results.

Several interdependencies exist among these items. For example, reporting by SCC implies that FuelTypes, but not RoadTypes will be distinguished. (The SCC itself includes an indication of both the fuel type and the roadway classification.)

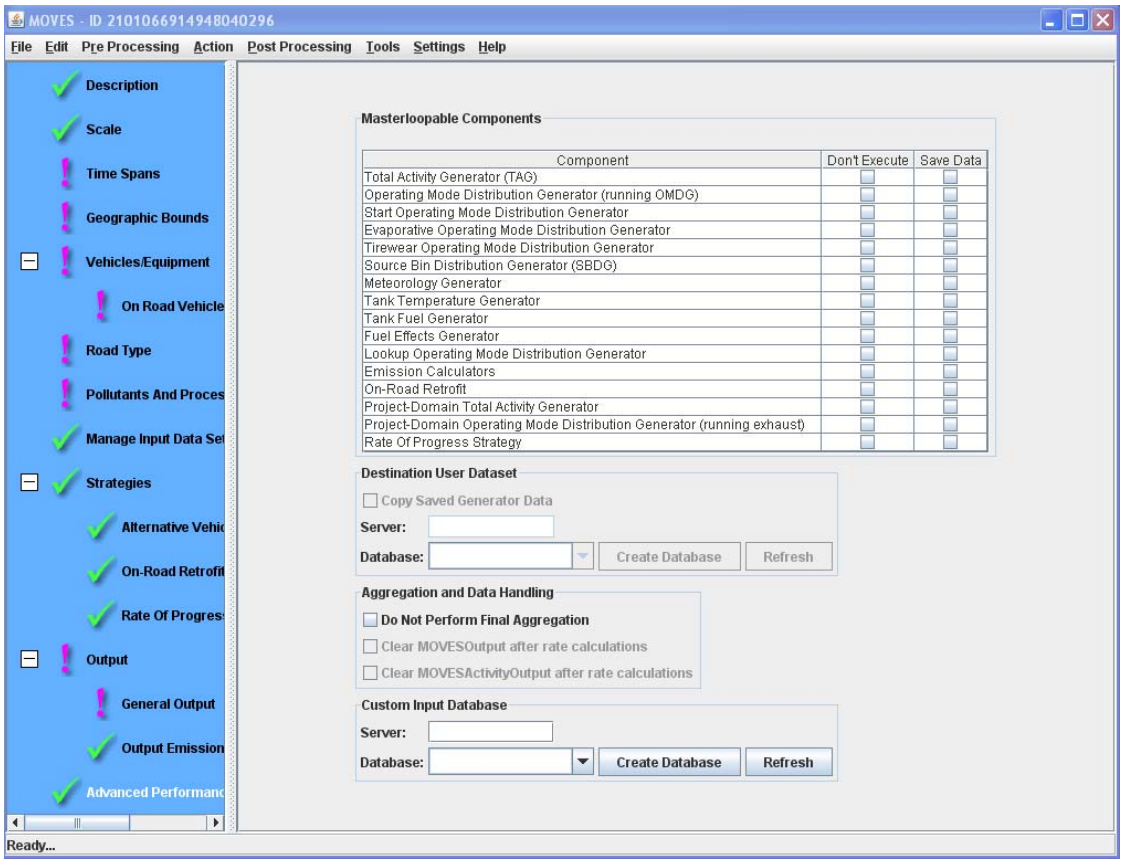
2.2.10.2.4 Estimate Uncertainty

The **Estimate Uncertainty** function is not available in MOVES2010.

2.2.11 Advanced Performance Features

The **Advanced Performance Features** panel invokes a variety of options. Some will be use to many users, others require specialized knowledge.

Many of these are 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. When this option is selected, the MOVES output tables may include rows with duplicate key fields. The results from these rows must then be summed to produce useful output. Use this feature with great care.

When the **Emission Rates** calculation type is selected on the **Scale** panel, the user may choose to "**Clear MOVESOutput after rate calculations**" and "**Clear MOVESActivityOutput after rate calculations.**" These options are recommended because the tables they affect may be quite large, and the values the tables hold represent only an intermediate step in the rates calculations rather than the real activity for that time and place.

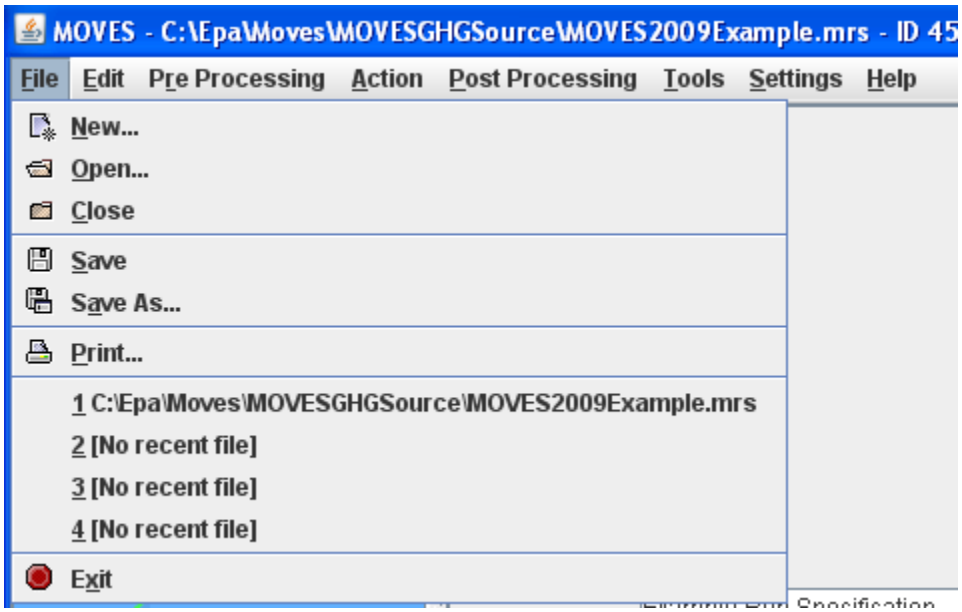
Finally, this panel includes a sub-panel where the user can specify an input database to be used instead of the MOVES default input database. Most users will not need this option.

2.3 Main Menu Bar

The Main Menu Bar runs across the top of the screen and contains eight drop down menus: **File, Edit, Pre Processing, Action, Post Processing, Tools, 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).

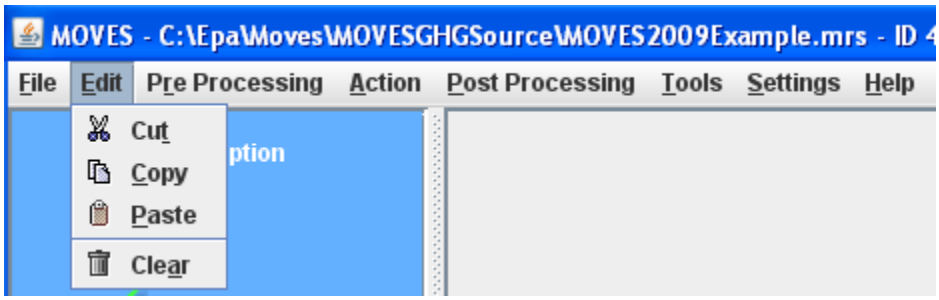
2.3.1 File

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). MOVES2010 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. Note that the Print command prints the XML version of the RunSpec, not a screen shot.



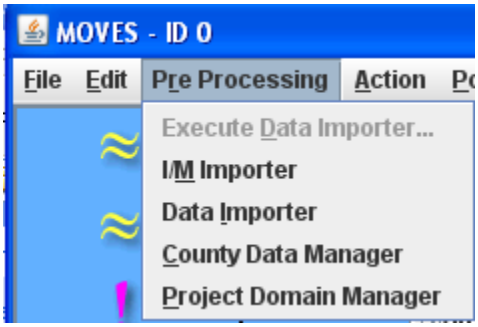
2.3.2 Edit

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.



2.3.3 Pre Processing

The **Pre Processing** menu contains four active options, **IM Importer, Data Importer, County Data Manager**, and **Project Domain Manager**. Each option enables the user to access importers for individual tables in order to create user-supplied databases for direct entry of data into MOVES. The **Project Domain Manager**, in particular, requires the user to provide virtually all of the data required to model a specific project.



The **County Data Manager** (CDM) and the **Project Domain Manager** (PDM)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 **I/M Importer** also provides users with the ability to define county-specifc data by allowing the user to edit the inspection and maintenance programs in an area. The **I/M Importer** can be used at any **Scale**.

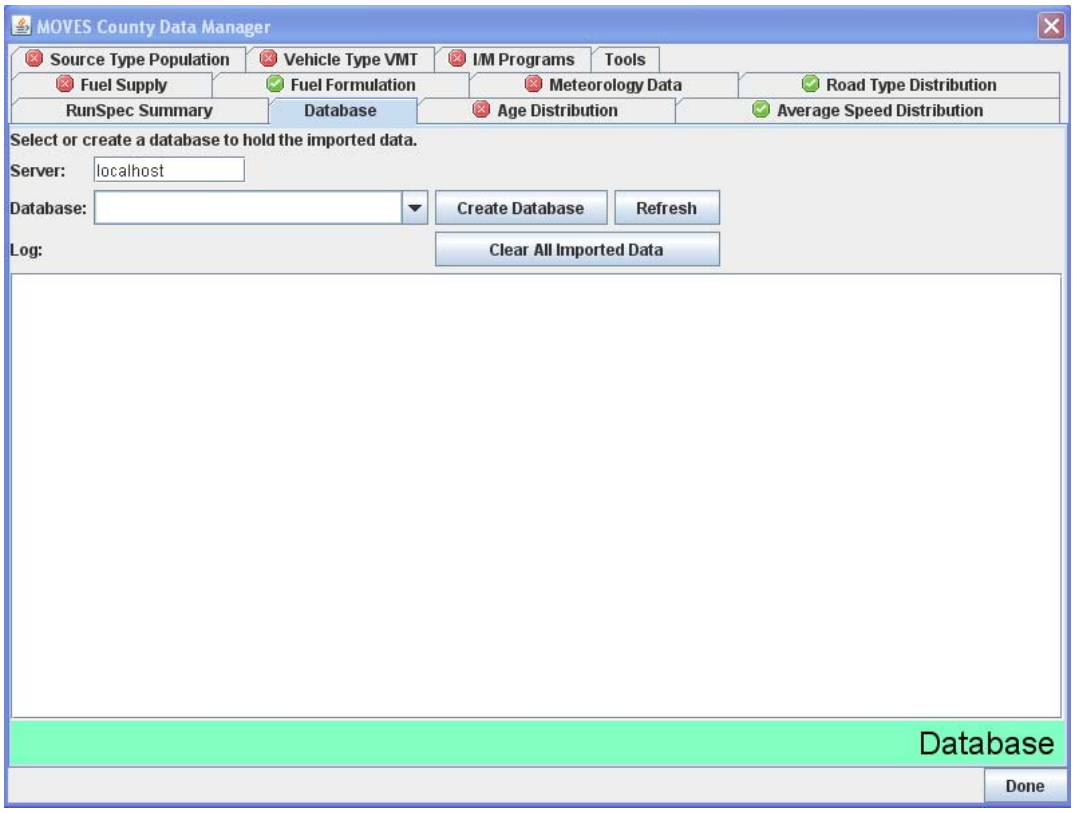
The **County Data Manager** and **Project Domain Manager** are designed to specify all the inputs required for certain kinds of runs and the **I/M importer** only allows I/M programs to be modified. The **Data Importer,** on the other hand, allows the user to select individual importers "ala carte" from the importer menu.

Caution! All tabs/fields in the **County Data Manager** and **Project Domain Manager** should 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, databases created with the **Data Importer** or **I/M Importer** should be input using the **Manage Input Data Sets** panel.

From the user perspective, there are three major differences between working with importers through the **Data Importer** and working with the same importers through the **County Data Manage**r and the **Project Domain Manager** (leaving the **I/M Importer** aside, for the moment). 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 identify the parameters where data is needed by displaying red "x-circles" on the importer tabs (see screenshot below)**.** Green checks indicate where data has been properly and completely imported. The **Data Importer** does not display such information.



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. The **Data Importer** can be used with any RunSpec.

Thirdly, the **Data Importer** is generally used to import data for a RunSpec at the **National Domain/Scale**. Therefore, when importing **Population** or **Vehicle Type VMT** at the **National Domain/Scale**, these inputs should be national values even if only a small number of counties are selected. MOVES will use the default allocation factors for the counties selected in the RunSpec to apportion data to each county.

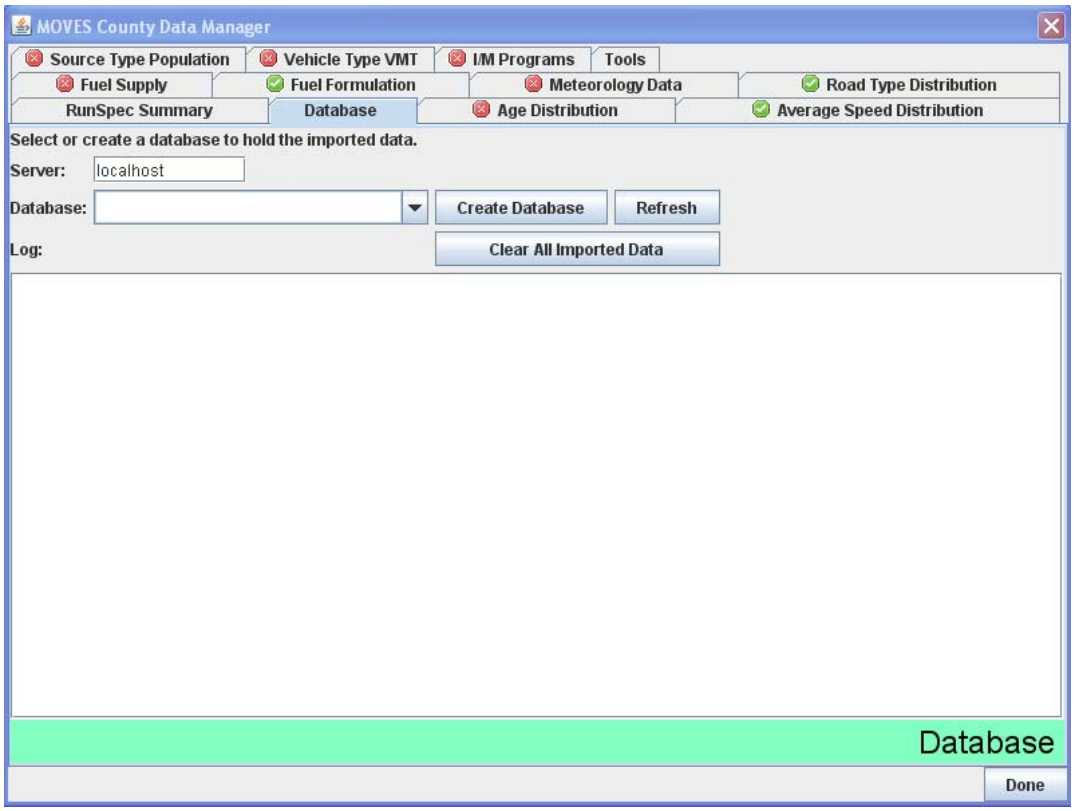
This user guide will use the **Data Importer** screens to explain functionality of individual importers. Specific instructions associated with using the **County Data Manager** or the **Project Domain Manager** to establish a new domain are included where appropriate. The sections below will also provide additional detail for the **I/M Importer**, as necessary.

When the user selects one of the data manager options, the manager 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.

2.3.3.1 Database

The **Database** tab is the default initial tab for the **Data Importer** menu option. It contains: a text box where the user can select an input database from a drop-down list or name a new database; three buttons to aid in the creation and management of the database; and a scrollable log detailing the success of the data import and other helpful diagnostics.

Note Data cannot be imported until an input 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 input 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, refresh the list of databases in the drop-down list, and clear the **Log**. The final button is the "**Clear All Imported Data**" button, which clears all the imported data (but not all the tables in the input tables as some are "core" tables that are populated based on the selections in the RunSpec at the time the database was created).

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

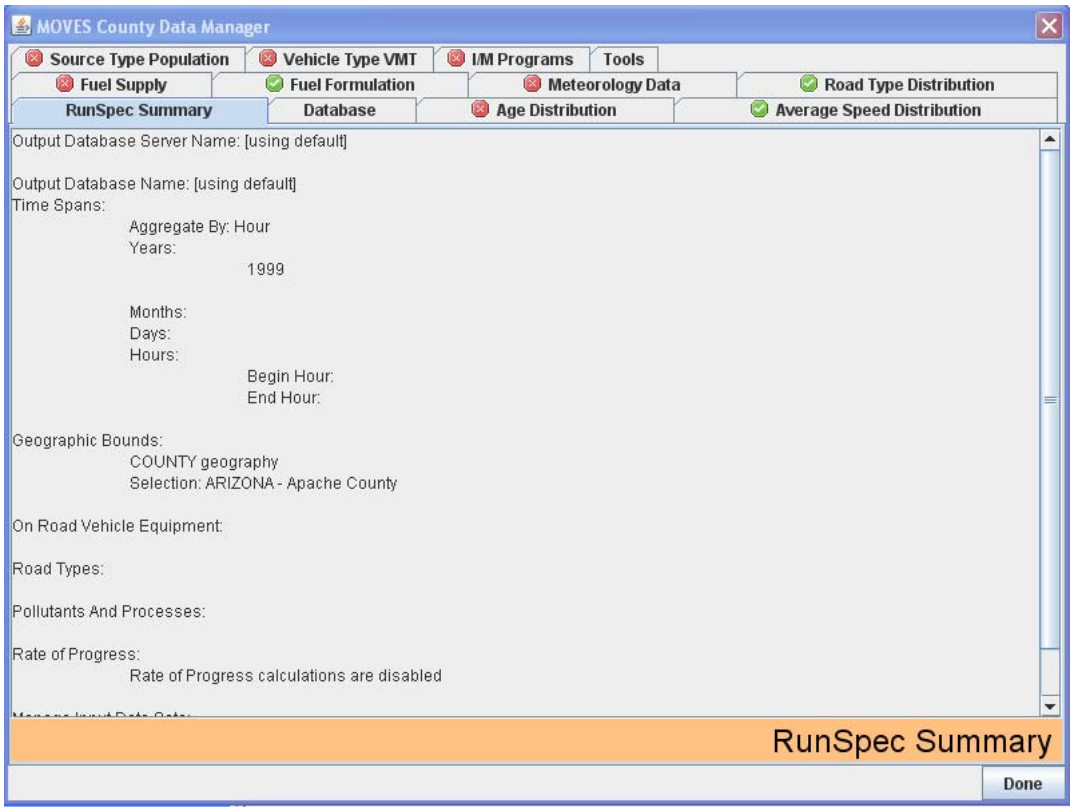
!Tip The input database should follow a naming convention to differentiate it from other databases; typically, using "_in" at the end of the database name is recommended.

Caution! When a database is created for a RunSpec at the **County** or **Project Domain/Scale**, a "year" table is automatically created with the year selected in the RunSpec. If any files are imported with a different year in a field, MOVES will add this year as an additional record in the"year" table. However, even if the incorrect data is cleared using the importer, the "year" table will not be changed, and MOVES will not execute. Users should take care to not import different years into an input database, but if this occurs there are currently two solutions: 1) Create and populate a new database with corrected files; or 2) Use MySQL to view and delete the row from the table. The MySQL queries to do this are: **SELECT * FROM** [database_in].year; **DELETE * FROM** [database_in].year **WHERE** yearID=[undesired_year];

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. The **Log** panel also includes the description of the imported file, if one was entered.

2.3.3.2 RunSpec Summary

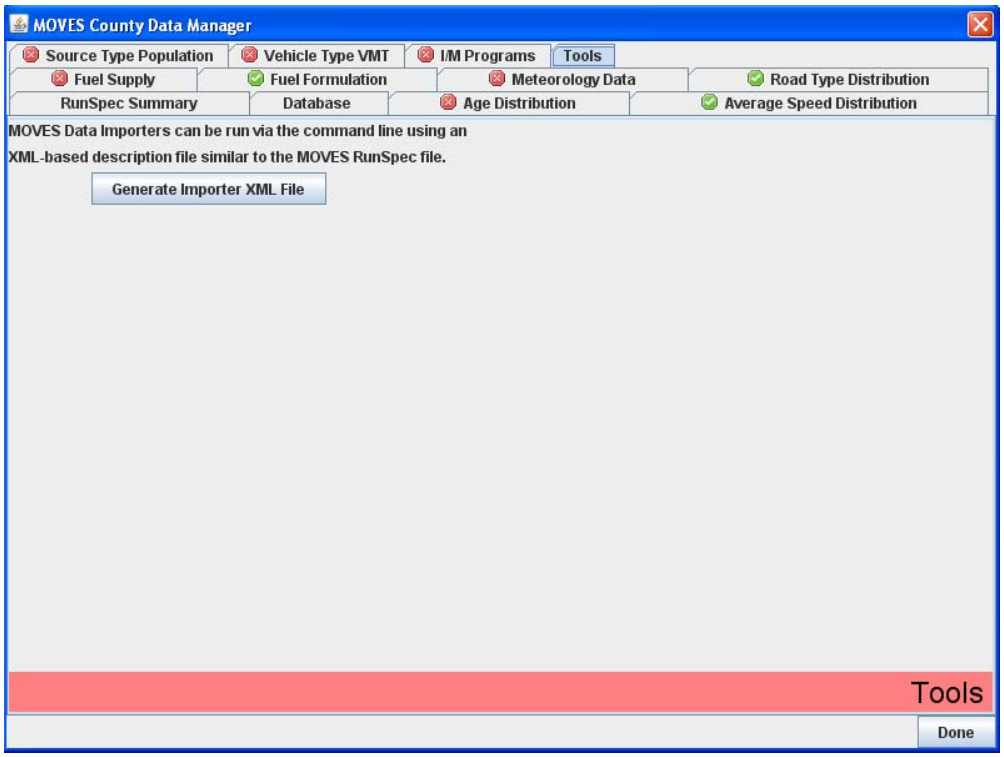
The **RunSpec Summary** provides a summary of RunSpec parameters defined by the user. The user cannot view the **Navigation** panels of the GUI without closing the **Importer**, but doing so may impact the user's ability to effectively employ "**Generate Importer XML File**" in the **Tools** tab. Therefore, 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 made in defining the RunSpec.



The **RunSpec Summary** panel 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 panel is auto-generated and does not accept user input.

2.3.3.3 Tools

The **Tools** tab is provided to facilitate batch input of imported data into the MOVES system, such as 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. Note, the XML file only includes files that have been imported during the active session so the XML must be generated before the user closes the **Data Importer.**



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; therefore, naming conventions are important to use this tool effectively. 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.

2.3.3.4 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 and ensuring it has the correct format. 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 "**Export Default Data**" button is used before fully defining the Run Spec.

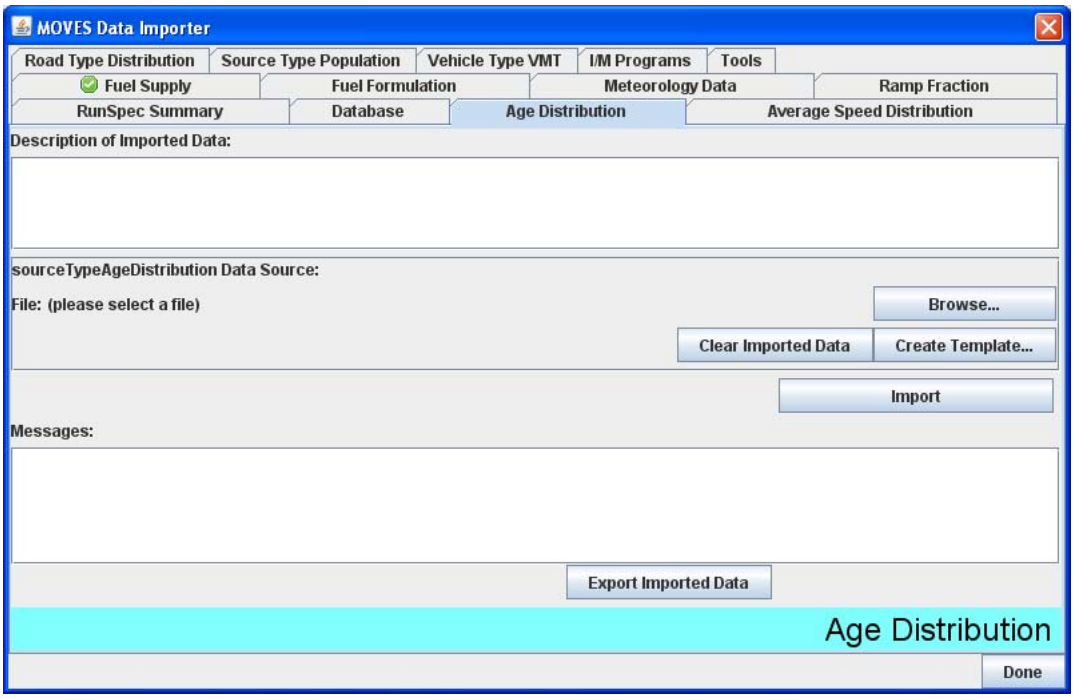
!Tip EPA has developed MOBILE6 converters to help users prepare import-ready data files from their existing MOBILE6 files. These converters are available on the MOVES website.

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 MOVES2010:

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
FuelFormulation	X	X	X
MeteorologyData	X	X	X
I/M Coverage	X	X	X
OperatingModeDistribution			X
LinkSourceType			X
LinkDriveSchedules			X
Off-Network			X

Table 2.3.3.4a

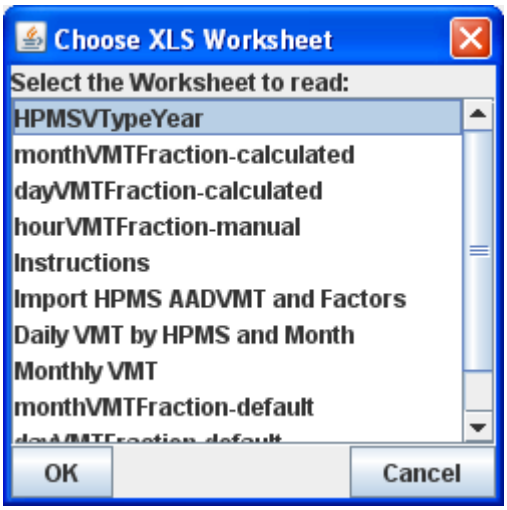
When the user selects one of these tabs, a panel opens allowing the user to define the import parameters. Each **Importer** tab is organized in the same way, with a place for the user's description of the data; a place where the corresponding MOVES default database table and the user's input file are identified; buttons to browse for data files, create templates, clear, import, and export data; and a space for messages informing the user of the success or potential errors with the import process.



Near the top of each impoter panel is a text box that allows users to enter a description of the data or file being imported. Any text entered here will be reported and saved in the Database Log. Just below the Description box is the Data Source section of the window. Along the left side of this section is text that states: [MOVESTable] Data Source:. The [MOVESTable] corresponds to the name of the table in the MOVES default database for which the tab will import data. In most cases, the tab header and MOVESTable name are similar, but in some cases they are not (e.g. **Meteorology Data** corresponds to the ZoneMonthHour table because temperatures are defined for each zone, month, and hour). Users can refer to the MOVESTable in the MOVES default database to learn more about that table, its structure, and the data contained within it.

Note The default database table may contain more fields than the template or exported default files (described below). There are several reasons why these additional fields may exist in the default table (e.g., the population and VMT tables each include a growth factor that is not used in the **County Domain/Scale** because only a single year can be modeled). Users should be aware that if the fields do not exist in the template or exported default data file, then these fields should not be added or populated.

Also in the Data Source section of the window are the **Browse**, **Clear Imported Data**, and **Create Template** buttons. After clicking the **Browse** button, the user can search through the computer's (or network's) drives to identify the file containing the data to be imported. Upon selecting a file, users may be prompted to select a worksheet:



The correct worksheet should be highlighted before the "OK" button becomes active.

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 (e.g., .xls for an Excel file) 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, as specified in the Run Spec. 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 MOVES2010.

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 (the latest fuelyear in MOVES2010 is 2012).

In many cases, users can import duplicate information by using "wildcards" in the input files. See Table 2.3.3.4b**.** 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 populate 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 1* - Applies to all motorcycles 2* - Applies to all passenger cars 3* - Applies to all passenger trucks 4* - Applies to all buses 5* - Applies to all single unit trucks 6* - Applies to all combination trucks
hpmsVTypeID	ALL
countyID	ALL *Note: Only one county is allowed in the CDM.*
zoneID	ALL

Table 2.3.3.4b

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.

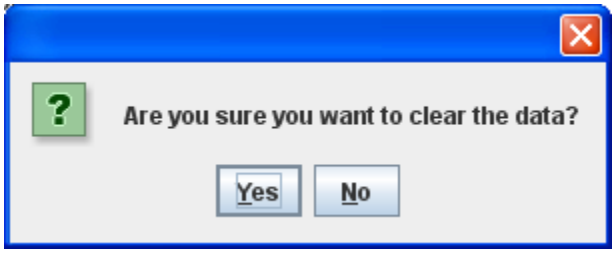
Note "Warning" messages do not necessarily mean that the imported data is incorrect. For example, users may receive a "Warning" if data for a source types, month, etc. are imported, but that source type or month is not selected in the RunSpec. On the other hand, "Error" messages indicate situations where the source file should be amended to correct the problem, the data should be cleared, and the file re-imported.

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, which is in the Data Source section of the window. 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. These exceptions 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 an "**Export**" button will cause the **Export [*Importer*] Data** window to open. The user defines the file path, file name and file extension in the **Export Data** window, and clicks the "**Save**" button, to export the data to the specified location.

Occasionally, clicking an "**Export**" buttons may result in the following pop-up window:

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

2.3.3.4.1 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.

2.3.3.4.2 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. While the MOVES table that this importer populates (SourceTypeYear) includes fields for sales growth rate and migration factor, neither of these fields should be imported. They are extraneous because the importer sets the selected year as a base year and defines the population.

Caution! Vehicle Type VMT, Source Type Population, and Age Distribution are the three tables that MOVES needs to define a base year. If any one of these tables is imported, then all three must be imported, otherwise MOVES may not generate output. However, this requirement does not apply to the **Project Domain/Scale**.

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.

2.3.3.4.3 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). The distribution of ageIDs (the variable for age) must sum to one for each vehicle type and year.

Caution! Vehicle Type VMT, Source Type Population, and Age Distribution are the three tables that MOVES needs to define a base year. If any one of these tables is imported, then all three must be imported, otherwise MOVES may not generate output. However, this requirement does not apply to the **Project Domain/Scale**.

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.

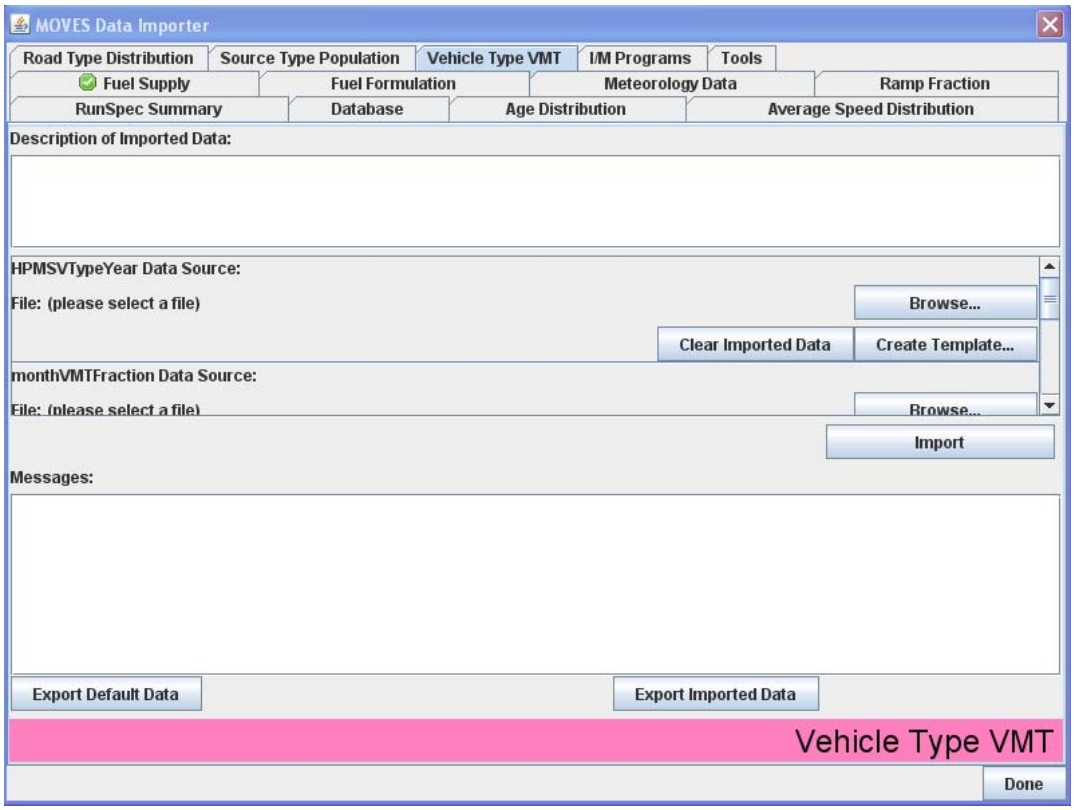
2.3.3.4.4 Vehicle Type VMT and VMT Fractions

The **Vehicle Type VMT** importer allows the user to enter yearly vehicle miles traveled (VMT) and the monthly, type of day, and hourly VMT fractions. These values will represent

national values for the general **Data Importer** and county-specific values for the **County Data Manager**. MOVES requires that "yearly" VMT be imported regardless of the time span selected for the run, although the VMT Fraction tables can be modified such that the value does not have to actually represent the VMT for the entire year in the area being modeled. Note that the appearance of this table differs from other tabs as multiple data sources are listed in the data source section. Users can expand the size of the CDM window or use the scroll bar to view all the data sources and buttons.

!Tip The MOVES website has spreadsheet tools to calculate yearly VMT from Average Annual Weekday VMT (AADVMT) and to convert MOBILE6 formatted VMT data into MOVES format.

Caution! Vehicle Type VMT, Source Type Population, and Age Distribution are the three tables that MOVES needs to define a base year. If any one of these tables is imported, then all three must be imported, otherwise MOVES may not generate output. However, this requirement does not apply to the **Project Domain/Scale**.



The "**Export Default Data**" button for this tab exports the default monthVMTFraction, dayVMTFraction, and hourVMTFraction tables. An HPMSVTypeYear table with values of zero for all HPMS classes is also exported, which can serve as a template that the user can edit before importing, but the user is required to acquire/generate the VMT 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.

Each of the VMT Fraction tables must sum to 1 across a varying set of fields. The monthVMTFraction table must sum to 1 for each sourcetype. The dayVMTFraction table must sum to 1 for each source type-month-road type combination. The hourVMTFraction must sum to 1 for each source type-road type-type of day combination. In the monthVMTFraction table, users must also correctly identify whether the year being modeling is (Y) or is not (N) a leap year in the 'isLeapYear' field.

Caution! For all the VMTFraction tables (month, day, and hour), a "Warning" message will appear if the fractions provided by the user sum to less than 1, and the tab header will erroneously turn to a green check. This should be an "Error" message, and the tab header should be red. Users must enter VMTFraction tables that sum to 1 even if not all 12 months, 2 types of day, or all 24 hours are being modeled.

A user wishing to model a portion of a year has two options: The user may import a true yearly VMT value (i.e. a value defining VMT for all months, days, and hours) and give the desired allocation to the time periods being modeled and populate the other months, day, or hours with placeholder values such that the each of the VMTFraction tables correctly sums to 1. Or a user may import only the amount of VMT appropriate for the time period being modeled and may assign values that correctly allocate this VMT over the modeled time periods.

The Import button will import the file identified for each data source at once. Therefore, users can, but are not required to, select all four files before importing. Users can also clear one data source file at a time by clicking the Clear Imported Data button associated with each data source.

2.3.3.4.5 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 combination of 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 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. See Section 2.3.3.4.12 for information on entering average speed information for Project level modeling.

2.3.3.4.6 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.

Note When comparing input roadtype distributions to the VMT output in the **MOVESActivityOutput** table, it may not appear that the road type distribution was applied properly if the user looks only at individual hourly or daily VMT results. However, users must compare the input road type distribution to the total weekly VMT by weighting the VMT results by the number of each type of day (5 weekdays, 2 weekend days). The VMT should be compared at a weekly level because of the effect of the dayVMTFraction table on the VMT output. For example, in the default DayVMTFraction table, the weekend fraction for rural roads is slightly higher than the weekend fraction for urban roads. If these default dayVMTFraction values are applied, the VMT in MOVES output by daytype will show a slightly higher-than-average fraction of total VMT on urban roads on weekdays, and on rural roads on weekends.

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.

2.3.3.4.7 Ramp Fraction Importer

The **Ramp Fraction Importer** allows the user to modify the fraction of ramp driving time 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.

For Inventory calculations, the **County Data Manager** automatically applies default values of 0.08 (8%) for this parameter if the user does not provide input, so importing ramp fractions is optional when **County Domain/Scale** is chosen.

Note The **Ramp Fraction** tab will only become available if an unrestricted road type (i.e. 2 or 4) is selected in the RunSpec.

For Rate calculations, which output running emissions by speed bin, the ramp fraction imported here is ignored. The ramp fraction is automatically set to zero (0) for all roadtypes in all Rate calculations.

2.3.3.4.8 Fuel Formulation Importer

The **Fuel Formulation** importer and the **Fuel Supply** importer should be used together to input appropriate fuel data. The **Fuel Formulation** importer allows the user to select an existing fuel in the MOVES database and change its properties, or create a new fuel formulation with different fuel properties. All fuel formulations imported in the **Fuel Supply** importer must exist in the MOVES default FuelFormulation table or be imported. If the user defines a new fuel, all of the fields except the last three (i.e., BioDiesel, Cetane and PAHContent) are required. Biodiesel may be modeled by entering and biodiesel volume in the Fuel Formulation table and a marketshare for the new biodiesel fuel in the Fuel Supply table. The inputs for the Cetane and PAHContent variables are currently inactive in MOVES.

The **Fuel Formulation** importer allows the user to change the following fuel properties.

Property	Description
FuelFormulationID	Fuel formulation identification number. Must be greater than 100 and less than 25000
Fuel Sub-Type	Fuel Sub-type coding—see Template File
RVP	Reid vapor pressure in psi
Sulfur Level	Fuel sulfur level in ppm Sulfur
ETOH Volume	Ethanol Volume (% vol)
MTBE Volume	MTBE Volume (% vol)
ETBE Volume	ETBE Volume (% vol)
TAME Volume	TAME Volume (% vol)
Aromatic Content	Aromatic content (% wt)
Olefin Content	Olefin content (% wt)
Benzene Content	Benzene content (% wt)
E200	Lower volatility percentage (%)
E300	Upper volatility percentage (%)
VolToWtPercentOxy	Constant based on oxygenate type
BioDieselEsterVolume	BioDiesel Ester Volume (%)
Cetane Index	Not Implemented - NULL
PAHContent	Not Implemented - NULL

Table 2.3.3.4.2

2.3.3.4.9 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 given fuel type (gasoline, diesel, etc.) must sum to one for each county, fuelyear (i.e., calendar year), and month. In MOVES2010, the user may select fuels from those available in the default database or create their own fuel inputs. Any new fuels which are added to the **Fuel Supply** importer must either be added in the **Fuel Formulation** importer or must already exist in the default MOVES database. Users must be sure to identify a fuel formulation for all vehicle-fuel type combinations selected in the **On Road Vehicle Equipment** panel

This importer also allows the user to supply a data element, "marketShareCV", which is the coefficient of variation for the market share. This would be used if uncertainty calculations were enabled. For MOVES2010 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 MOVES2010 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.

2.3.3.4.10 I/M Importer

The **I/M Importer** allows the user to import information describing inspection and maintenance programs. When defining I/M programs in an area, users should refer to the supporting information tables to identify the numeric codes for the pollutant-process, source type, and test standard for the program being described. I/M programs can only be applied to gasoline vehicles (fueltypeID of 1) and only annual or biennial inspection frequency should be used (inspectFreq of 1 and 2, respectively).

CAUTION! I/M emission effects are not currently available for diesel fuel or continuous I/M inspection frequency.

A field that requires more detailed explanation is the IMProgramID. This identifier is an arbitrary number that is serves as a substitute for unique combinations of test standard, test frequency, begin model year, and end model year. Therefore, the same IMProgramID can be associated with multiple source types and pollutant-processes, but not with different test standards, frequencies, or model year ranges.

Note Only one I/M program can apply to each pollutant-process, source type, fuel type, model year combination. If the user imports a file that has programs that overlap for any of these parameters, an "Error" message will notify the user that there are "duplicate active programs". The user can resolve the "Error" message by either ignoring one of the programs that created the error or modifying the programs so that they do not overlap.

For the I/M tab header to be changed from a red X to a green check, the user must account for all the default I/M programs in some way. This can be done by simply importing the defaults; however, if the user believes the default information is not correct, then a file with modified data should be entered. The preferred method for entering modified data is to enter a value of "N" for No in the "UseIMYN" field for all the default I/M programs for which the user will provide new data. Then the user should add to the file new entries with "Y"s in the "UseIMYN" field and new IMProgramID values; this approach ensures that all the default programs are accounted for and the user has full control of the new programs. Another method is to attempt to edit the default programs, but then the user runs a higher risk of creating programs with the same IMProgramID, but different test standards, frequencies, or model year ranges, and thus a higher likelihood of introducing errors.

Note ASM and IM240 programs can only apply to 1981 and later model years; Exhaust and Evaporative On-Board Diagnostics (OBD) programs can only apply to 1996 and later model years.

The complianceFactor is a value (0 to 100 scale) that defines the percentage of vehicles that participate in the I/M program. Users should consult the MOVES Technical Guidance for more information.

The "**Export Default Data**" button provides I/M coverage data for many programs throughout the United States, but users should review the default programs as current local data may be more accurate and up-to-date.

Note This is the only importer tab in the **I/M Importer**, although that tool also has the **Database**, **RunSpec Summary**, and **Tools** tabs.

2.3.3.4.11 Zone Road Activity Importer

The **Zone Road Activity Importer** is used only if the **Custom Domain** option is chosen in the **County Domain Manager**. Currently, MOVES only has the capability to model one zone at a time within a Custom Domain. The Zone Road Activity tab erroneously shows a green check, but data MUST be imported for MOVES to calculate results properly; a value of 1 should be entered for the SHOAllocFactor for each road type so that all of the VMT input by the user is assigned to the custom domain.

CAUTION! The green check for this importer is not working correctly; when modeling a Custom Domain, user data entry is required.

This tab does not have the "**Export Default Data**" button, so users should create a template to populate the SHOAllocFactor field with a value of 1 for each road type..

2.3.3.4.12 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 its MOVES 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). Other required inputs for each link are countyID, zoneID, and roadTypeID (these same entries 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, the average speed of all of the vehicles on the roadway link in the given hour, and the average road grade of a particular link. The link description field is a text field included for reference and has no impact on the calculations.

In addition to roadway links, 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, link average speed and link average grade (percent grade).

For Project level modeling, users have a choice of ways to describe the speed, acceleration and power of the vehicles being modeled. Depending on the information available for each roadway link, a user may enter an operating mode distribution, a drive schedule or an average speed. For a given roadway link, a user-supplied operating mode distribution input will take calculational precedence over an imported drive schedule, which will take calculational precedence over an average link speed input when more than one is entered for a given link.

Use of a link average speed input in the **Links Importer** requires the least amount of road link data to be provided by the user. All that is required is an average link speed and an average link road grade (in units of percent grade). The model will use built-in driving schedules and an interpolation algorithm to produce a default operating mode distribution.

Table 2.3.3.4.12 shows the allowable range for the average speed input in Project level by source type. The user may enter any speed in units of miles per hour (mph) within the minimum and maximum range as the average speed input for a given link. Average link speed inputs outside of the minimum and maximum speed range may produce no output records for the given link. No warning or error message is provided.

These average speed ranges exist in MOVES project level because these ranges represent the extent of the available driving cycle data in the MOVES model. The MOVES model does not extrapolate beyond its built-in data. Should the user desire to model a link with an average speed that is less than the minimum or greater than the maximum, they must supply the second by second driving schedule or an equivalent operating mode distribution.

Allowable Average Speed Input Range for Project Level Input By Source Type			
sourcetypeid	**sourcetypename**	**Minimum Speed (mph)**	**Maximum Speed (mph)**
11	Motorcycle	2.5	73.8
21	Passenger Car	2.5	73.8
31	Passenger Truck	2.5	73.8
32	Light Commercial Truck	2.5	73.8
41	Intercity Bus	4.6	72.8
42	Transit Bus	4.6	72.8
43	School Bus	4.6	72.8
51	Refuse Truck	2.2	71.7
52	Single Unit Short-haul Truck	4.6	72.8
53	Single Unit Long-haul Truck	4.6	72.8
54	Motor Home	4.6	72.8
61	Combination Short-haul Truck	5.8	71.7
62	Combination Long-haul Truck	5.8	71.7

Table 2.3.3.4.12

2.3.3.4.13 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 driven 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. If the user enters data for source types which are not selected in the Runspec, MOVES will ignore that data. The Project level calculator will not re-normalize the fractions to omit the contribution of source types which are not selected in the Runspec.

2.3.3.4.14 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 each have a distinct emission rate. 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 either the **Link Drive Schedules Importer** is not used, or the link average speed input is not entered in the **Links** Importer**. Operating Mode Importer data** has precedent over data entered in the **Link Drive Schedules Importer** and the **Links Importer** if conflicting data are entered**.**

2.3.3.4.15 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 used**.** For a given roadway link, an operating mode distribution input will take calculational precedence an imported drive schedule. An imported drive schedule will take calculational precedence over an average link speed input when more than one is entered for a given link. However, at least one of three (an operating mode distribution, a link drive schedule or a link average speed) must be entered for each of the user's defined roadway links.

2.3.3.4.16 Off Network Importer

The **Off Network Importer** is used only in the **Project Domain Manager**. It provides information about vehicles which 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 during the hour being modeled. 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.

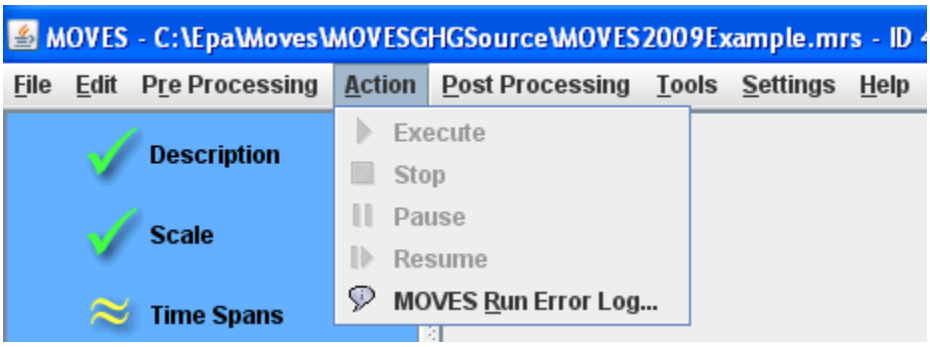
For example, a vehicle population of 10 long-haul combination trucks with a start fraction of 0.5, an extended idle fraction of 0.25 and a parked vehicle fraction of 0.5 implies that the project includes the emissions associated with 5 starts, 150 minutes of extended idling (10 vehicles * 60 minutes *0.25) and 300 minutes of parking (10 vehicles * 60 minutes * 0.5) for this sourcetype.

Note MOVES2010 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 MOVES2010 does not compute evaporative emissions at the Project Domain/Scale; thus, the "parked vehicle fraction" is ignored.

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 **Execute** 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.

Note Note that, depending where the MOVES program is in its processing, the Stop command may not end a MOVES run immediately. If necessary, MOVES can be terminated by clicking the X in the upper right hand corner of the MOVES GUI or the MOVES command line window.

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.

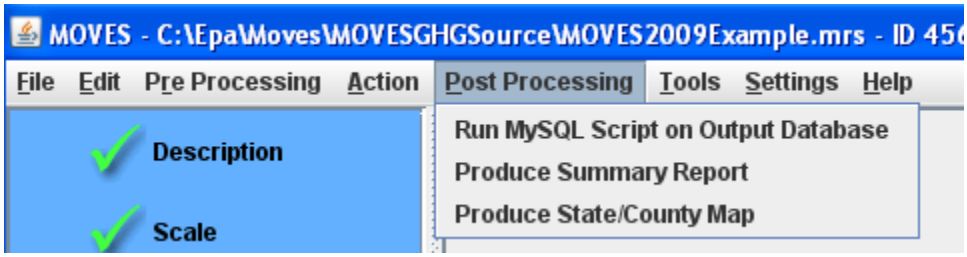
Note The **MOVES Run Error Log** option is available only if a MOVES Output database has been created.

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 message. 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. 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 MOVES2010. 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.

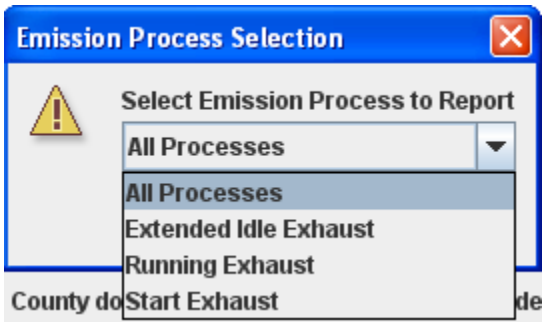
The second script "**TabbedOutput.sql**" takes three MOVES MySQL output tables (MOVESOutput, MOVESActivityOutput and MOVESRun) and converts them into tab-delimited text files that can be read by a spreadsheet program such as Excel or Lotus123. This is useful if the user does not want to work with the results in the MySQL relational database format. In MOVES2010, this can also be done with the **Summary Reporting** feature. Both scripts work with **Inventory** output, not **Emission Rate** output.

!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 a user-created script 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.

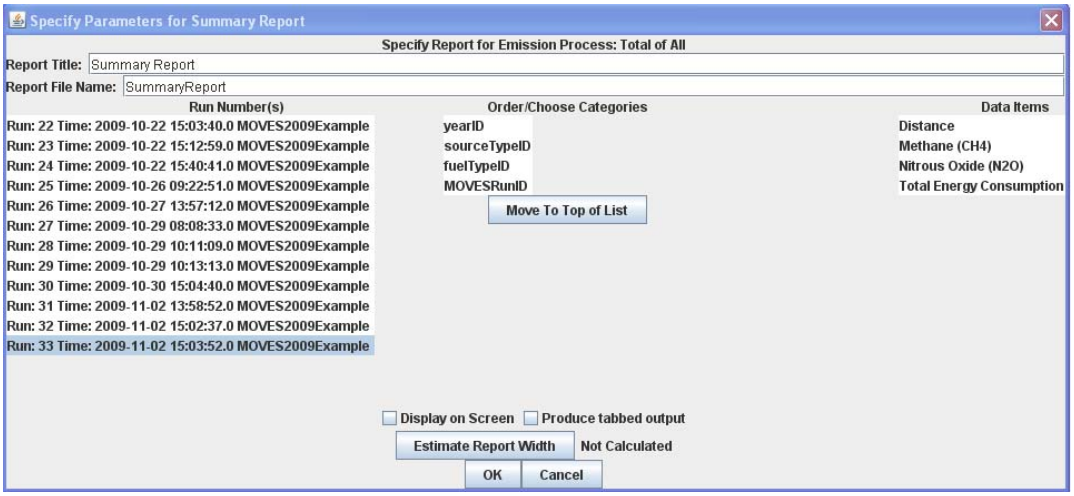
2.3.5.2 Produce Summary Report

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.



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.

1. 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.

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.

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.

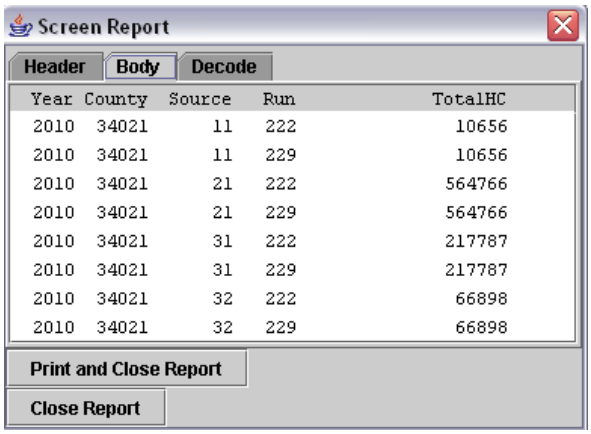
5. Select the data items to be included in the report. At least one data item besides "**Distance**" must be selected.

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. Note that choosing the Display on Screen option may cause MOVES to fail, if the selection criteria yields an extremely large set of data.

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). In this case, Summary Reporter should not be used to display results for a month or year, since the current code erroneously simply sums these two days together. The user should distinguish by dayID in the report and then externally 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).

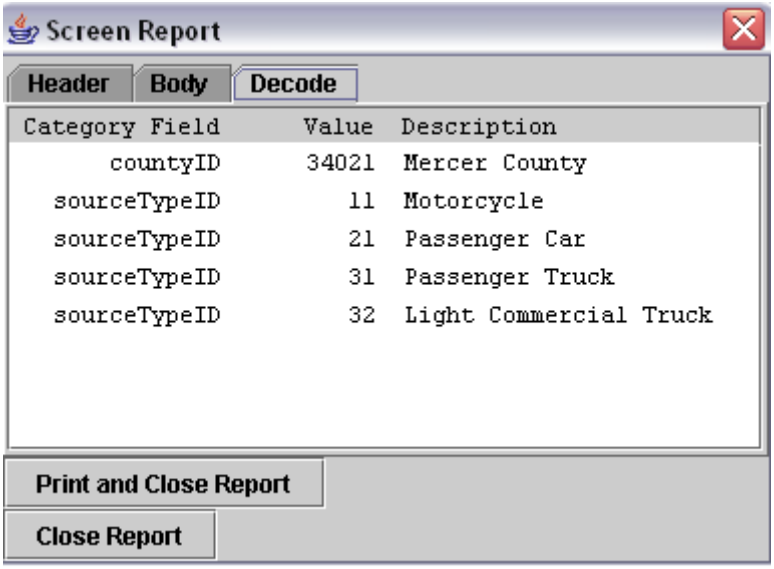
Because the width of a report can be a constraining factor (for example, when printing the report) the "**Estimate Report Width**" button can be used to estimate of the width of a report with the currently selected items. Printed report output is obtained by producing a screen report and then choosing to print.

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



Note The Screen Report shown here is not consistent with the "Specify Parameters" panel shown above, but it does illustrate the type of results a user can expect to see.

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.



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.

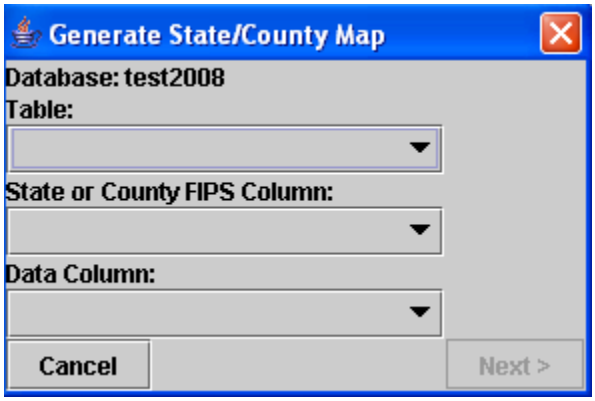
Caution! Both buttons close the report. The Summary Reporter does not store reporting options, other than the last report title and base file name used. Because information is not remembered between Summary Reporter runs, the user should not close the report until the user is done viewing it.

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 with very specific input requirements. 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 that points to the same output file, needs to be loaded in MOVES. When the user chooses the **Produce State/County Map** menu option, 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.)

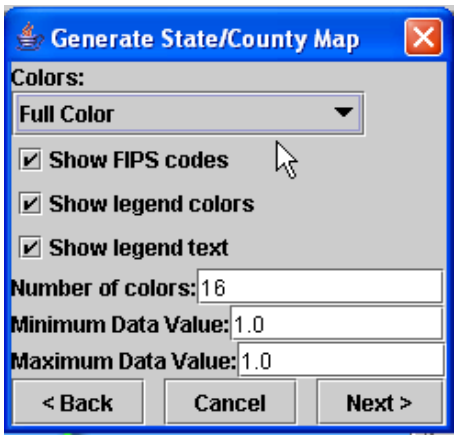


To view RunSpec output on a map, the user must first choose a table contained in the output database that has "**CountyID**" or "**StateID**" as a column and associates each emission quantity with each county or state selected only once; in other words, if there are multiple entries for the same pollutant in the same county, then the **Produce State/County Map** tool cannot be used. 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 and save it into the output database indicated by the currently loaded RunSpec.

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,

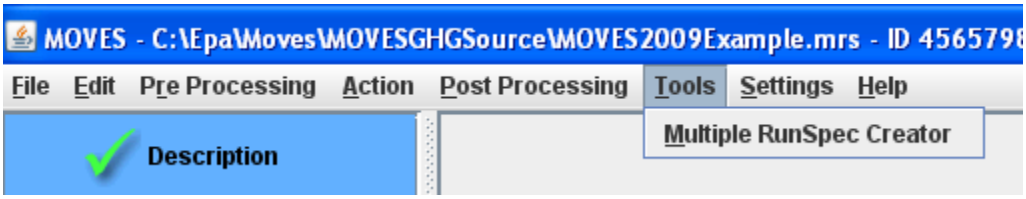
the user instructs the mapping tool to use results from the RunSpec to generate the map output. After making selections in this window, the user should click "**Next**".

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 Tools

Tools provides a drop-down menu with the choice **Multiple RunSpec Creator.**



This menu item contains a list of MOVES tools. In MOVES2010, the only tool on the list is the **Multiple RunSpec Creator**, which creates a set of new MOVES runspec files using the currently loaded runspec and a user-supplied control file as a template.

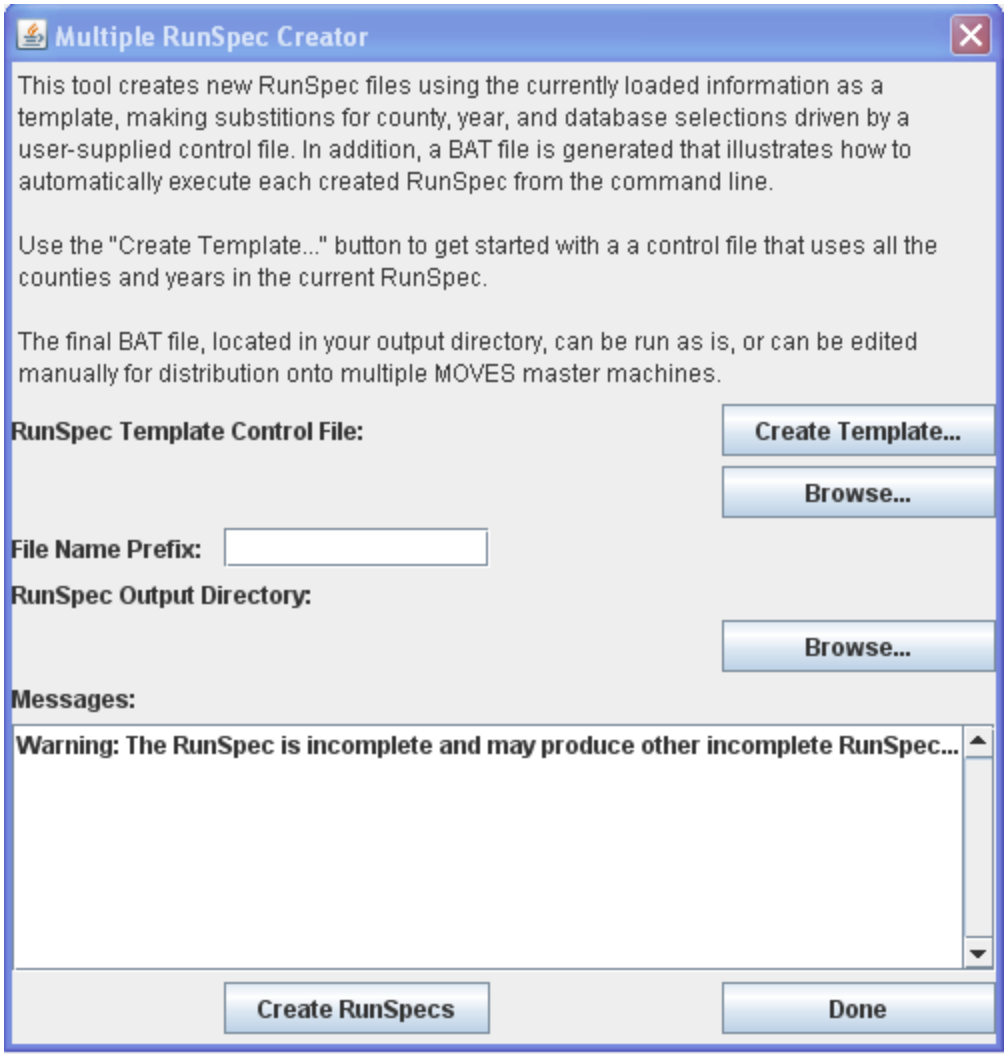
2.3.6.1 Multiple RunSpec Creator

The **Multiple RunSpec Creator** creates a set of new MOVES RunSpec files using the currently loaded RunSpec information, and additional information in the form of a user-supplied control file. The control file is a list of county, year and database data selections in the form of a Microsoft Excel spreadsheet file (.xls). The tool produces a set of MOVES RunSpecs and a batch (.BAT file) file that can execute the MOVES RunSpecs from the DOS command line (MOVES batch files cannot be executed from inside the MOVES GUI).

Before using the **Multiple RunSpec Creator**, the user should load or create a viable (all green check marks) Runspec (the tool will accept an incomplete RunSpec, but will subsequently

generate a set of incomplete RunSpecs which require further manual editing). The **Multiple RunSpec Creator** also requires the selection of at least one county and one year in the RunSpec.

If the menu item **Multiple RunSpec Creator** is selected, the "**Multiple RunSpec Creator**" panel opens. The top of the panel contains some reminders about the purpose of the tool and mentions the output .BAT file. Error messages and other feedback to the user are provided in the messages area at the bottom of the window.



Once the panel is opened, use the "**Create Template…**" button to get started with a Control File that uses all the counties and years in the current RunSpec. The **Create Template** feature will fill the Control Filecf with the information available in the RunSpec. However, the user may edit the Control Filecf and add additional rows and columns of information. The user may add additional counties, years and database names. For each row in the Control Filecf MOVES will create a new Runspec.

The figure below shows a template Control File (an .XLS in this case) that will create RunSpecs for a list of counties, each with a different output database:

	A	B	C	D	E	F	G	H	I
1	CountyID	County Description	Year	Additional Text Name	Primary MOVES DB	Output DB	Domain DB	Advanced Features DB	User DB
2	26161	MICHIGAN - Washtenaw County	2009	Washtenaw	MOVESDB20090112	WashtenawOutput			
3	26163	MICHIGAN - Wayne County	2009	Wayne	MOVESDB20090112	WayneOutput			
4	26161	MICHIGAN - Washtenaw County	2010	Washtenaw	MOVESDB20090112	WashtenawOutput			
5	26163	MICHIGAN - Wayne County	2010	Wayne	MOVESDB20090112	WayneOutput			
6	26161	MICHIGAN - Washtenaw County	2011	Washtenaw	MOVESDB20090112	WashtenawOutput			
7	26163	MICHIGAN - Wayne County	2011	Wayne	MOVESDB20090112	WayneOutput			

!Tip If the *National* SCALE is selected, a set of runspecs will be generated for unique combinations of county, year and additional text name. Information from the other columns will be inserted into the runspecs. If the *County* or *Project* SCALE are selected, a set of runspecs will be generated only for unique combinations of county and year. Note, in all three cases, the user must separately create the required input databases.

The Control File fields are:

Field	Notes
CountyID	Mandatory. The FIPS ID of the county or custom domain (generic county) in the format 99999
County Description	Describes the county. This is not mandatory but is helpful as it gets stored within each RunSpec. It will be the label for the generic county if the Custom Domain option is used.
Year	Mandatory. A year contained within the current MOVES default database, typically 1990 and 1999-2050 inclusive.
Additional Text Name	Optional, 20 characters max. Additional text to be included in both the file name of a RunSpec file and in the RunSpec's description (which gets stored to the output database during simulation). Since this field is used as part of a file name, it cannot contain operating system special characters such as & : / \ * ?
Primary MOVES DB	Name of the primary MOVES database to be used with the RunSpec. If blank, the created RunSpec will use the "Custom Input Database" (as set on the Advanced Performance Features panel) of the current RunSpec, and if that is blank the created RunSpec will use the MOVES default database configured at the time it is executed. This input will never change the name of the Default Database in the MOVES Configuration File as shown in the pull down menu item "Settings – Configure MOVES".
Output DB	Name of the database to store the created RunSpec's output. If blank, the created RunSpec will use the output database of the currently loaded RunSpec, and if that is blank an error message will be shown and processing stopped.
Domain DB	When using County or Project domains, MOVES requires an input database specific to the domain. If blank, the model will use the database named in the RunSpec. If neither is available, an error message will be shown and processing stopped.
Advanced Features DB	If the "Copy Saved Generator Data" checkbox is marked on the Advanced Performance Features panel, this field specifies the database where the data should be saved.
User DB	The MOVES GUI allows users to supply their own custom data using the Manage Input Data Sets panel. On that panel, zero or more user databases are listed. Such databases can be specified in the Control File in the User DB column and the columns to the right of it. For example, if two databases are desired, fill the User DB column with the first database, and fill the next column with the second.

	MOVES places no limit on the number of such user databases.

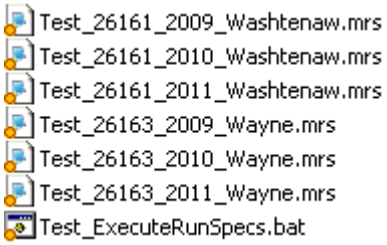
After successfully creating a template file, populating the file and saving it as a Control File, the user should return to the **Multiple RunSpec Creator** panel and browse for and select the Control File. The name of the Control File will be echoed in the Multiple RunSpec Creator panel to the left of the first Browse button.

The next step is for the user to enter a file name prefix in the File Name Prefix box. This input is a mandatory, 150 character limit, field that gives a prefix to the file name of each created RunSpec and .BAT file. Good usage of this field, perhaps with a project number or metropolitan area name, will help preserve the usefulness of the created RunSpecs.

After setting a prefix, use the second Browse button for an output directory to hold the created RunSpecs and .BAT file. Remember that many files may be created, so it may be best to create a new directory to hold the new files.

The Create RunSpecs button will check input values (control file name, file prefix, and output directory), read the control file, report errors, and if there are no errors, create the BAT and RunSpec files. Files are only generated if there are no errors.

As an example, using the above control file and a prefix of "Test", the files created are:



As can be seen, the naming convention is:

<prefix>_<countyID>_<year>[_<additional text>].mrs

<prefix>_ExecuteRunSpecs.bat

Note If RunSpecs are for the same county and year, it is important that the additional text be unique, otherwise, not all listed RunSpecs will be created.

The resulting BAT file from the example file is:

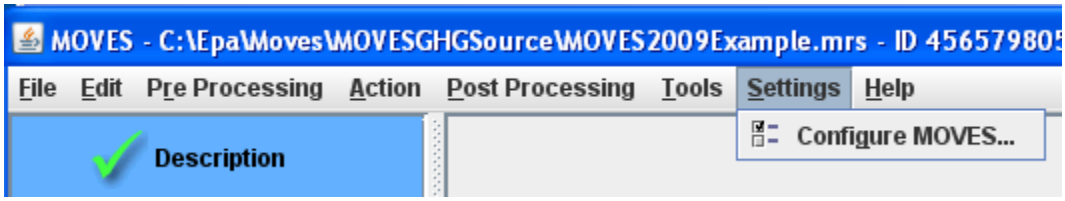
```
@echo off
rem Script generated by the MOVES Looping Tool
rem Based on control file:
C:\EPA\MOVES\MOVESGHGSource\Task910LoopingTool\ControlFile.xls
rem ------------------------------------------------------------
rem Command Line Notes:
rem "-e DEBUG" logs all types of messages to MOVESbatch.log
rem ------------------------------------------------------------
echo Changing to the MOVES folder and compiling code...
C:
cd "C:\EPA\MOVES\MOVESGHGSource"
call setenv.bat
```

```
call ant compile
rem ------------------------------------------------------------
echo Running Test_26161_2009_Washtenaw.mrs
java gov.epa.otaq.moves.master.commandline.MOVESCommandLine -e DEBUG -r
"C:\EPA\MOVES\MOVESGHGSource\Task910LoopingTool\Test_26161_2009_Washtenaw.mrs"
echo Running Test_26163_2009_Wayne.mrs
java gov.epa.otaq.moves.master.commandline.MOVESCommandLine -e DEBUG -r
"C:\EPA\MOVES\MOVESGHGSource\Task910LoopingTool\Test_26163_2009_Wayne.mrs"
echo Running Test_26161_2010_Washtenaw.mrs
java gov.epa.otaq.moves.master.commandline.MOVESCommandLine -e DEBUG -r
"C:\EPA\MOVES\MOVESGHGSource\Task910LoopingTool\Test_26161_2010_Washtenaw.mrs"
echo Running Test_26163_2010_Wayne.mrs
java gov.epa.otaq.moves.master.commandline.MOVESCommandLine -e DEBUG -r
"C:\EPA\MOVES\MOVESGHGSource\Task910LoopingTool\Test_26163_2010_Wayne.mrs"
echo Running Test_26161_2011_Washtenaw.mrs
java gov.epa.otaq.moves.master.commandline.MOVESCommandLine -e DEBUG -r
"C:\EPA\MOVES\MOVESGHGSource\Task910LoopingTool\Test_26161_2011_Washtenaw.mrs"
echo Running Test_26163_2011_Wayne.mrs
java gov.epa.otaq.moves.master.commandline.MOVESCommandLine -e DEBUG -r
"C:\EPA\MOVES\MOVESGHGSource\Task910LoopingTool\Test_26163_2011_Wayne.mrs"
```

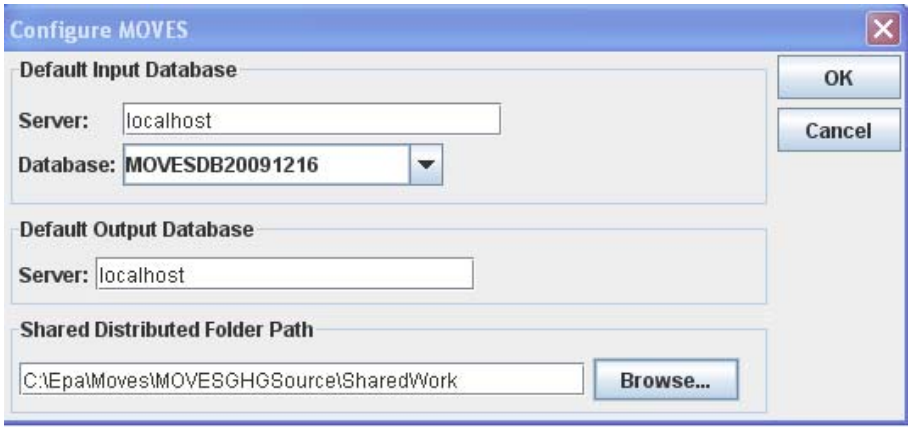
This .BAT file can be run as is (via the command line or by double-clicking), or can be edited manually for distribution onto multiple MOVES master machines.

2.3.7 Settings

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



2.3.7.1 Configure MOVES



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 desired server, although the user may input an alternate. 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 the default database distributed with MOVES2010. The user also can select a server to house the default output database server under the **Settings→Configure** menu option; "localhost" is appropriate in most cases. The actual output database file is named on the **General Output** panel to specify output parameters as part of the MOVES RunSpec setup process.

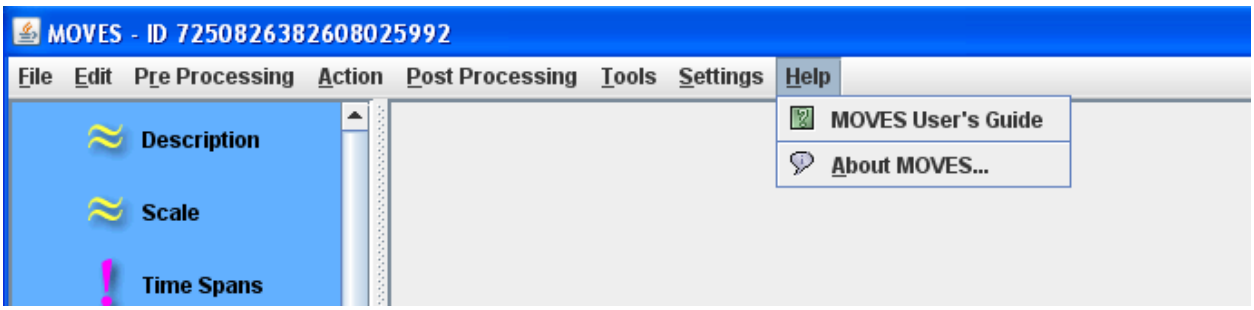
Caution! If the **Default Input Database** is changed the program should be stopped and restarted. This is because the user interface program will have already connected to the previously specified database in order to construct the GUI displays; thus the GUI may not match the new input database.

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.

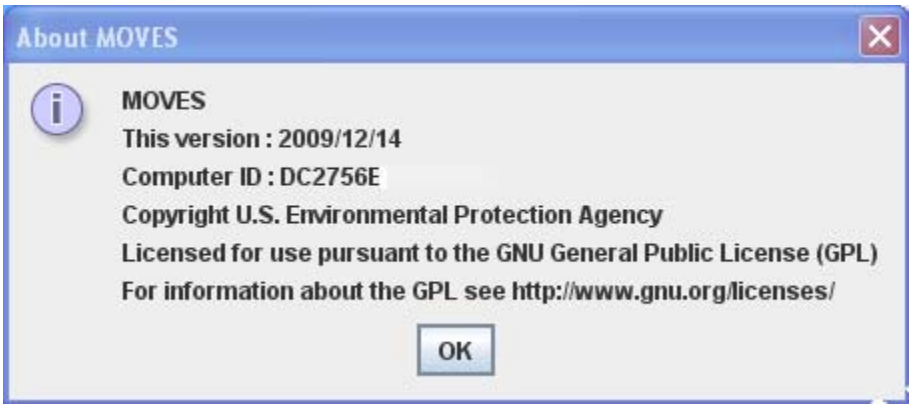
2.3.8 Help

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



Choose **MOVES User Guide** to open the document 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/.


About MOVES
MOVES
This version : 2009/12/14
Computer ID : DC2756E
Copyright U.S. Environmental Protection Agency
Licensed for use pursuant to the GNU General Public License (GPL)
For information about the GPL see http://www.gnu.org/licenses/
OK

Appendix A – Using MOVES to Generate Lookup Tables

MOVES can calculate emission inventories (in total quantity of emissions for a given time) or emission rates (emissions divided by distance or population), depending on the user selection of "**Inventory**" or "**Emission Rates**" in the "**Calculation Type**" section of the "**Scale**" panel.

When modeling a single time and place, the emission rate calculations in MOVES are significantly more time consuming and require more computer memory than inventory calculations. However, for large scale projects, a user may want to use the emission rate calculations to create a lookup table of rates that--though a post-processor--can be applied to many times and places, thereby reducing total MOVES run time. Successful application of this approach requires a clear understanding of the rates calculations in MOVES and careful planning. EPA is developing tools to automate the use of this feature to create inputs for EPA's Sparse Matrix Operator Kernel Emissions (SMOKE) model. This section describes the process more generally.

Defining a Scenario

A scenario defines the set of conditions to which the emission rates may be applied. A scenario typically defines a specific fuel mix and set of Inspection/Maintenance (I/M) programs. It also assumes a certain pattern of trip activity (number of trips per vehicle per day and the distribution of those trips in time).

Some characteristics of the scenario depend on user choices: If a given age distribution is part of the scenario definition, that is, if the lookup table is to be applied only to areas with one particular age distribution, the lookup output can be smaller because the user does not need output by model year. On the other hand, detailing output by model year allows users to apply the lookup table to areas with diverse age distributions.

A scenario generally does not imply specific temperatures or speed distributions. Instead, these characteristics are key fields in the output used to match the emission rate with the appropriate vehicle activity.

The Emission Rate Output Tables

Selecting **"Emission Rates"** on the **Scale** panel populates three additional output tables: **RatePerDistance**, **RatePerVehicle** and **RatePerProfile**. The **RatePerDistance** table stores emission rates for emissions that occur while a vehicle is in regular operation. The RatePerVehicle and RatePerProfile tables store emissions that occur while a vehicle is stationary, on "off network" roadtypes. The latter two tables differ because the emissions in the RatePerVehicle depend on the hour and the temperature while emissions in the RatePerProfile table also depend on temperatures in previous hours.

1) RatePerDistance: Includes emissions for the processes that occur while vehicles are operating: running exhaust (includes emissions during normal idle at traffic signals, etc), tire wear, brake wear, evaporative permeation, evaporative fuel vapor venting, evaporative fuel

leaks, crankcase running exhaust, refueling displacement and refueling spillage. The hour, day and month will not be relevant for most uses because the emissions are normalized with regard to activity. For national and county level runs, roadtype and avgspeedbin will be relevant. For project-level runs, linkID will be important.

Fields:

MOVESScenarioID
MOVESRunID
yearID
monthID
dayID
hourID
linkID
pollutantID
processID
sourceTypeID
fueltype ID
model year ID
roadtype ID
avgSpeedBinID
temperature
relhumidity
RatePerDistance

2)RatePerVehicle: Includes emissions for most processes that occur while vehicles are stationary : start exhaust, start crankcase, permeation, liquid leaks, and extended idle (long haul combination trucks only). The month will not be relevant for most uses since, in rate calculations, it serves primarily as an identifier for the fuel mix and will be redundant with MOVESScenarioID, but daytypeid and hourid are important key fields since start and other activity vary per vehicle per hour.

Fields:

MOVESScenarioID
MOVESRunID
yearID
monthID
dayID
hourID
zoneID
pollutantID
processID
sourceTypeID
fueltype ID
model year ID
temperature
RatePerVehicle

3) RatePerProfile: Includes emissions from the vapor venting process when vehicles are stationary. The daytypeid, hourid and temperatureprofileid are important key fields because the vapor venting emissions vary depending on activity and previous temperatures.

Fields:

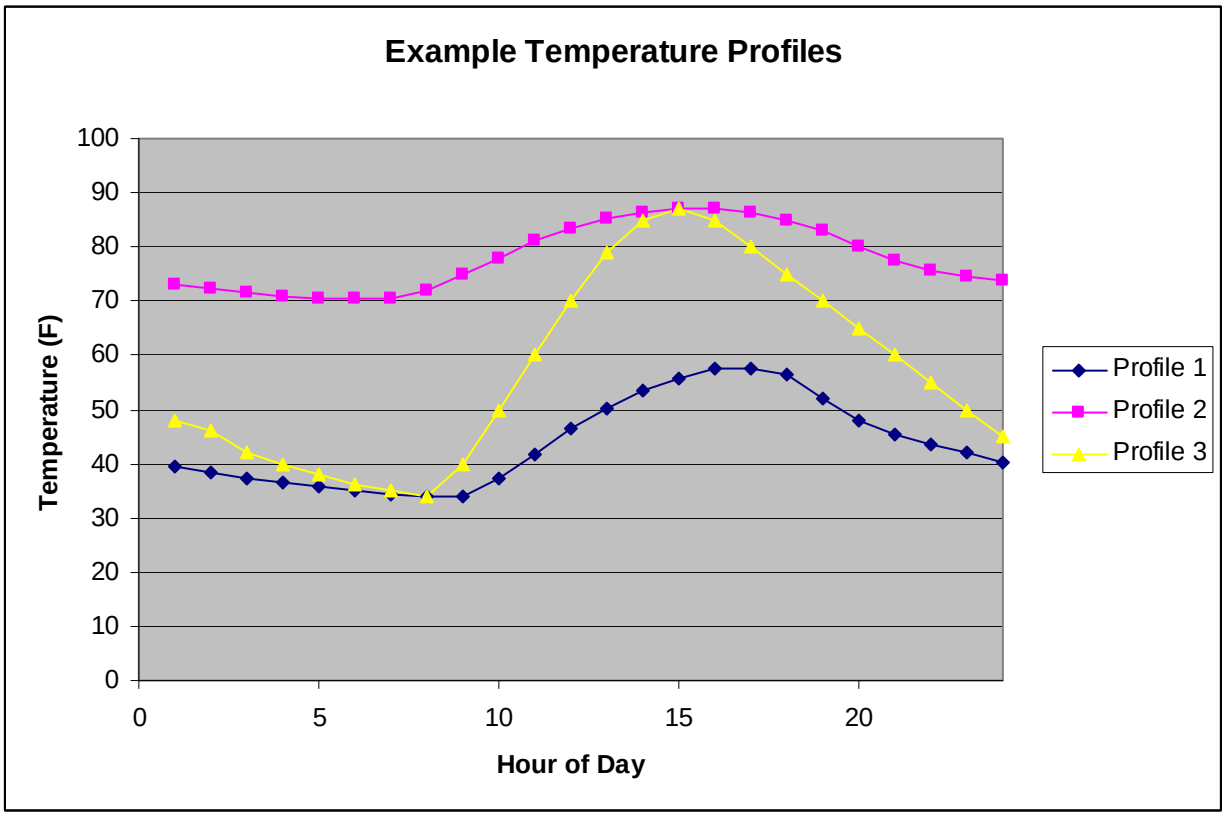
MOVES Scenario ID
MOVESRunID
TemperatureProfileID
yearID
monthID
dayID
hourID
pollutantID
processID
sourceTypeID
fueltypeID
modelyearID
temperature
RatePerVehicle

Setting Up Runs to Generate a Lookup Table

Setting up an emission rate lookup table for a single scenario requires a number of steps.

1) Define the scenario, and determine the counties, months, years and day-types in which it applies. Using the County Data Manager, a user can set up a run for the desired county, year, fuel supply, age distribution and I/M program.

2) Determine the temperatures and temperature profiles to be covered by the lookup table. A "temperature profile" is a set of 24 temperatures over the course of a day. If the user is modeling NOx, it may also make sense to include relative humidity as a variable.



3) Determine which pollutants and processes are of interest. Note that the modeling of hydrocarbons is more complicated than the modeling of other pollutants due to the need to include evaporative processes, and thus the need to represent temperature profiles as well as hourly temperatures.

4) Organize inputs to generate the most rates in the shortest total runtime, while maintaining a manageable output size. If multiple similar calculations are required, it is generally a good idea to combine them in the same run. However, due to the different requirements for different emission processes, redundant calculations can often be avoided by splitting the task into multiple runs.

For running exhaust, crankcase running exhaust, and refueling, hour doesn't matter, so 24 temperatures can be modeled as different hours in a single month If more than 24 distinct temperatures are to be modeled, additional temperatures can be modeled using additional months (as long as the fuel characteristics are set to be the same for each month). If this approach is used, the month field in the output essentially becomes a label for a group of temperatures, and does not represent an actual calendar month.

For start, crankcase start exhaust, liquid leaks and evap permeation, hour and temperature matter. Sets of temperatures can be entered as individual "months" Again, this creates output where "month" represents a group of temperatures rather than a calendar month. In the illustration above, the three temperature profiles could be modeled in a single run as three nominal months, one for each profile. Note, however, that Profile 3 shares an hour/temperature point with the each of the other profiles (in hour 8 and hour 15). For a lookup table with greater scope, it could be efficient to design the runs to eliminate such redundancy.

Vapor venting emissions depend on hour, temperature and previous temperatures. For these emissions we must model each profile as a separate month. Thus, within a given MOVESScenario, monthid becomes equivalent to temperatureprofileid. In some cases, it may be appropriate to "bin" similar temperature profiles and to apply the results from one profile to a group of similar profiles.

Temperatures (and humidities) can be entered into the MOVES ZoneMonthHour table, using MYSQL commands or through the County Data Manager. See User Guide section **2.3.3.4.1 Meteorology Data Importer** for more information on entering this data.

CAUTION! When producing Rates table for evaporative emissions, you must include at least one "running" emission process in the run specification because there are evaporative emissions that are reported only in the RatePerDistance table, but this table is populated only when a running emission process is selected. We plan to improve this in future versions of MOVES.

!Tip If "**Emission Rates**" are chosen on the **Scale** panel, output should be differentiated by "**Source Use Type**". Doing so allows VMT, Road Type Distribution, and Average Speed Distribution to become placeholders (i.e., they must still be imported, but their values do not impact the results); however, if output is not differentiated by source type, the emission rate becomes a weighted average of the source types selected in the RunSpec and the three inputs mentioned will impact the results.

Running MOVES to generate Lookup Tables

MOVES can generate lookup table output using the menu-driven interface, but for large-scale modeling, users will probably want to automate execution of a series of runs. See **Appendix C** for information on running MOVES from the command line. See **Section 2.3.6.1 Multiple Run Spec Creator** for information on a tool for creating a set of similar run specifications.

Applying MOVES Emission Rate Output

To calculate an emissions inventory from emission rate output, rates in RatePerDistance values need to be multiplied by the appropriate Vehicle Miles Travelled (VMT), and rates in RatePerVehicle and RatePerProfile must be multiplied by appropriate vehicle population values. Doing this correctly can require a number of considerations.

- Rather than simply summing all activity in a speed or temperature bin, it is usually appropriate to do an interpolation as the rates are applied. For example, to calculate the total emissions from 100 miles of VMT with an average speed of 17.6 mph, a simplistic approach would be to multiply all of the VMT by the emission rate for speed bin 5 (17.5 mph-22.5 mph), however this approach is overly sensitive to even very small changes in speed. To reduce these boundary issues, users should instead interpolate between speed

bins (in this example, between the rates for speed bin 4 (12.5-17.5) and speed bin 5) based on the average speeds for those speed bins. .

- Depending on how the runs were organized, the user should take care not to double-count emissions with duplicate emission rates (for example, if the three profiles illustrated above were each modeled as independent months, there would be duplicate values in the ratepervehicle table for 8am at 34F, and for 3pm at 87F). A simple join of the rates and population table using hour and temperature would create extra rows that the user would need to be careful not to include in a sum.

- The user should be aware that evaporative emissions are stored in all three tables, and all three need to be used to fully account for evaporative emissions that occur during both driving and parking. The rates in RatePerDistance need to multiplied by the appropriate VMT, and the rates in RatePerVehicle and RatePerProfile need to be multiplied by the appropriate population. They can then be summed to calculate the total emissions.

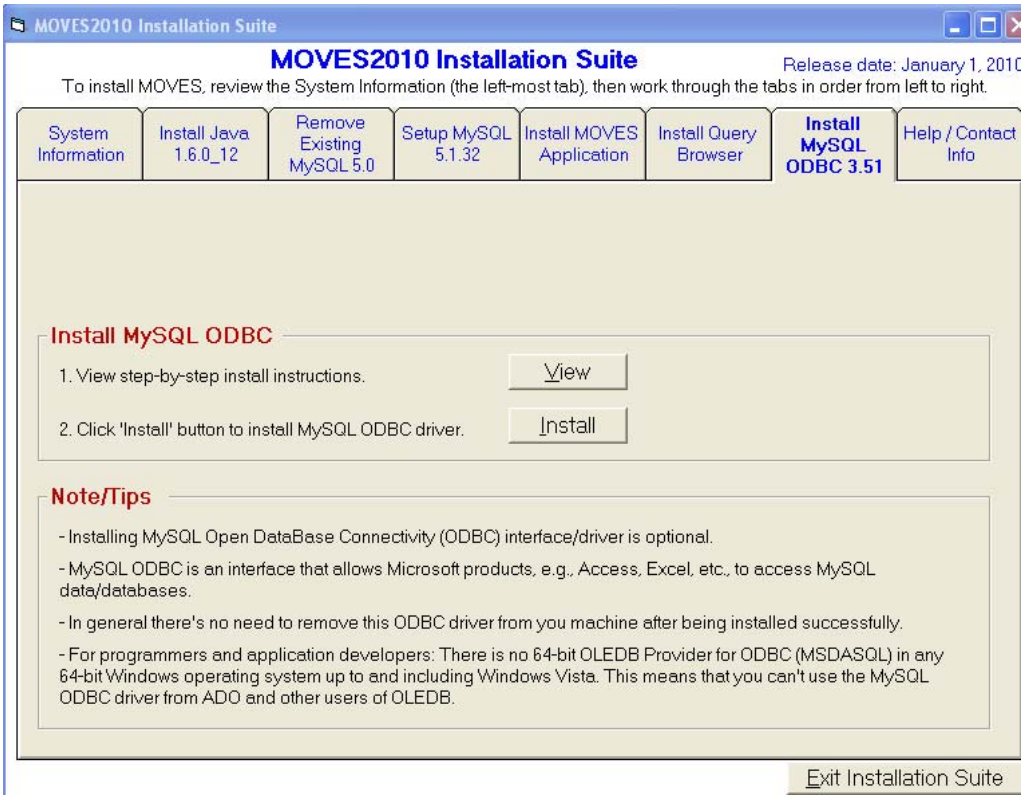
[This page intentionally left blank.]

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.

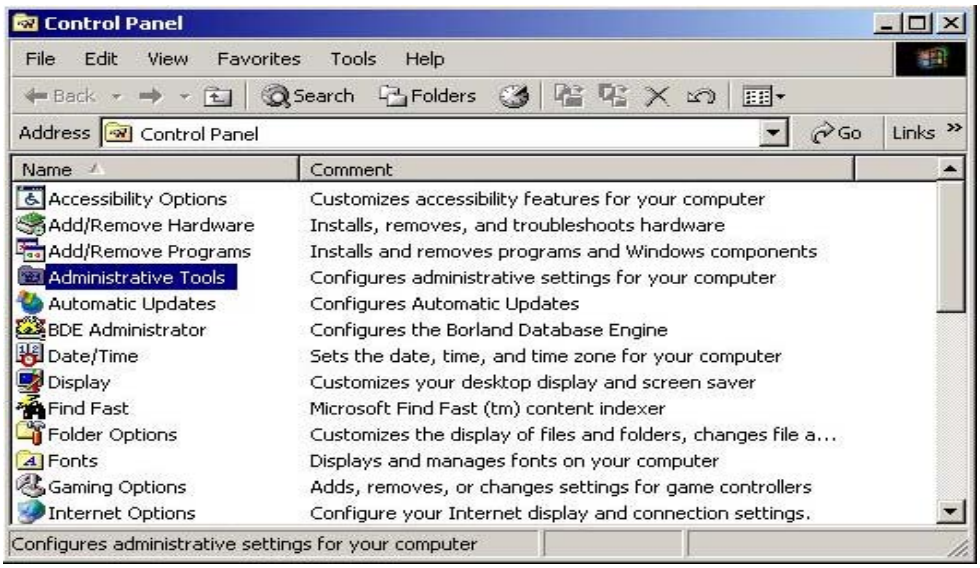
Adding a MySQL database as an ODBC User Data Source

1. Install MySQL driver, MyODBC-3.51.06.exe. This is part of the normal MOVES installation process and can be done by executing the MOVES2010 Installation Suite

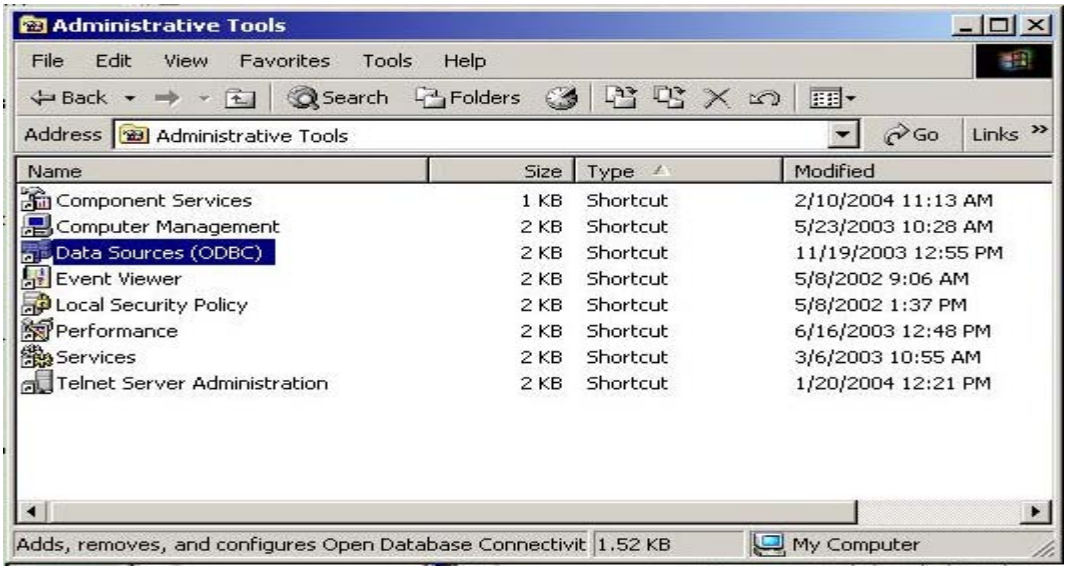


2. Create an ODBC data source with the same name as the folder which corresponds to the MySQL database.

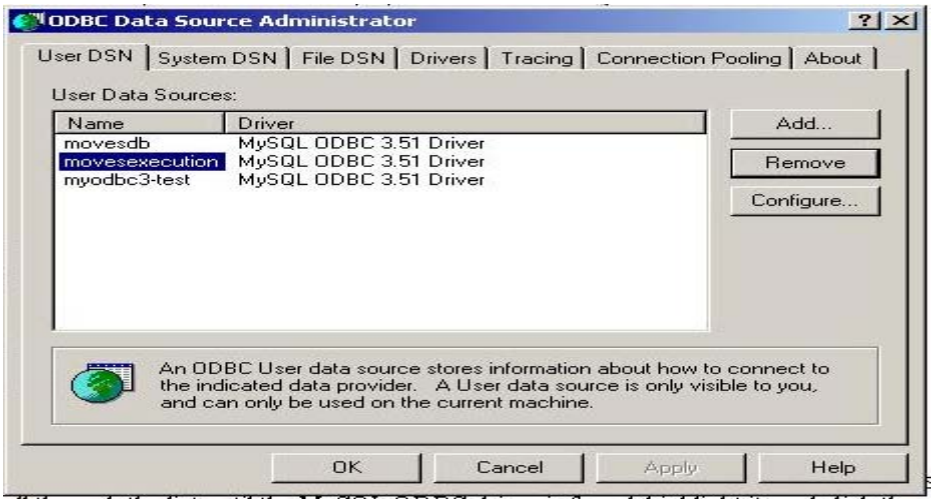
   a. In the Control Panel, select "Administrative Tools":


Control Panel
File Edit View Favorites Tools Help
Back Search Folders
Address Control Panel
Go Links
Name Comment
Accessibility Options Customizes accessibility features for your computer
Add/Remove Hardware Installs, removes, and troubleshoots hardware
Add/Remove Programs Installs and removes programs and Windows components
Administrative Tools Configures administrative settings for your computer
Automatic Updates Configures Automatic Updates
BDE Administrator Configures the Borland Database Engine
Date/Time Sets the date, time, and time zone for your computer
Display Customizes your desktop display and screen saver
Find Fast Microsoft Find Fast (tm) content indexer
Folder Options Customizes the display of files and folders, changes file a...
Fonts Displays and manages fonts on your computer
Gaming Options Adds, removes, or changes settings for game controllers
Internet Options Configure your Internet display and connection settings.
Configures administrative settings for your computer

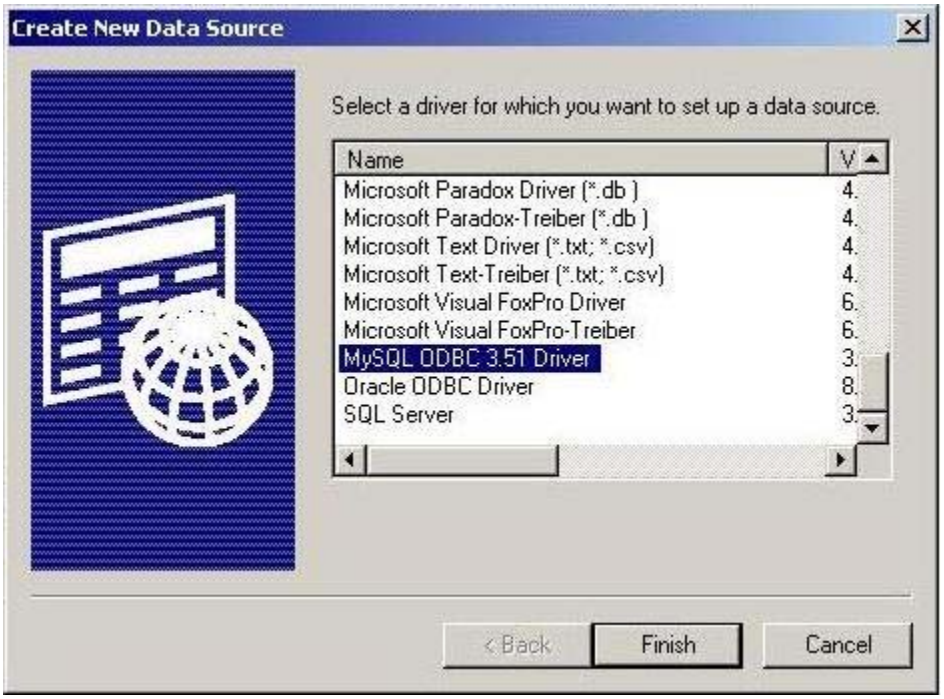
b. In the "Administrative Tools" menu, select "Data Sources (ODBC)":



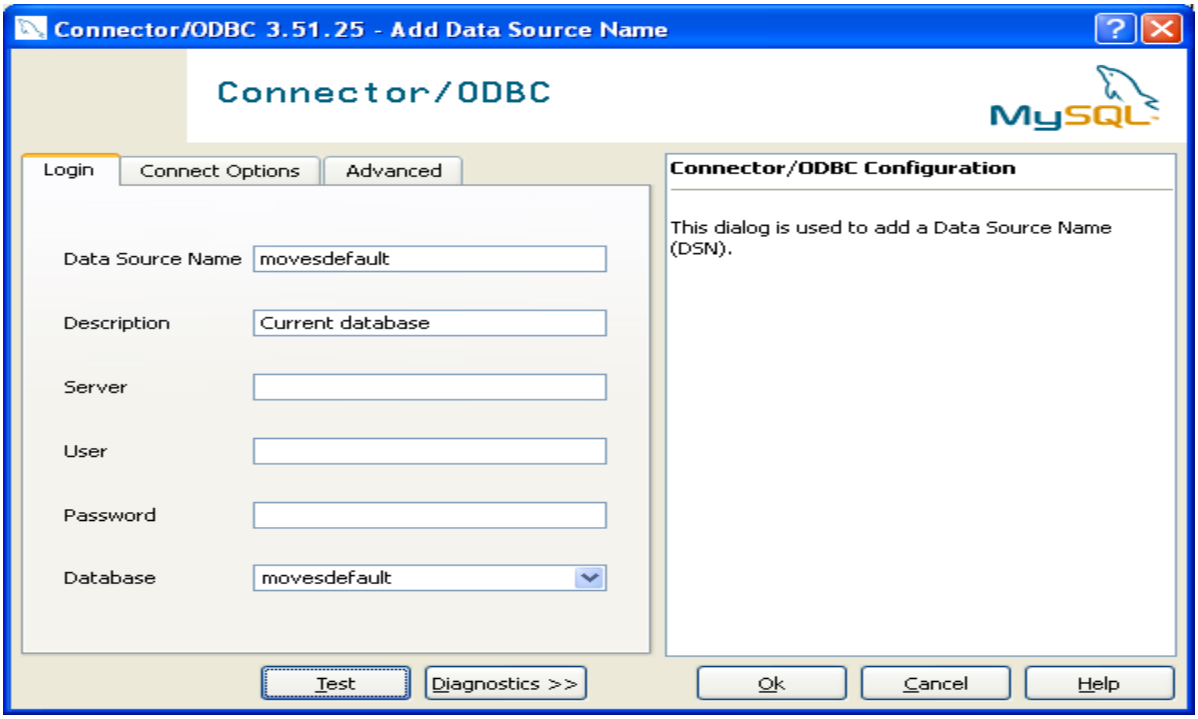
c. In the ODBC Data Source Administrator dialog choose to "Add" a "User Data Source."



d. The "New Data Source" should use the MySQL ODBC Driver that was installed earlier. Scroll through the list until the MySQL ODBC driver is found, highlight it, and click the "Finish" button.

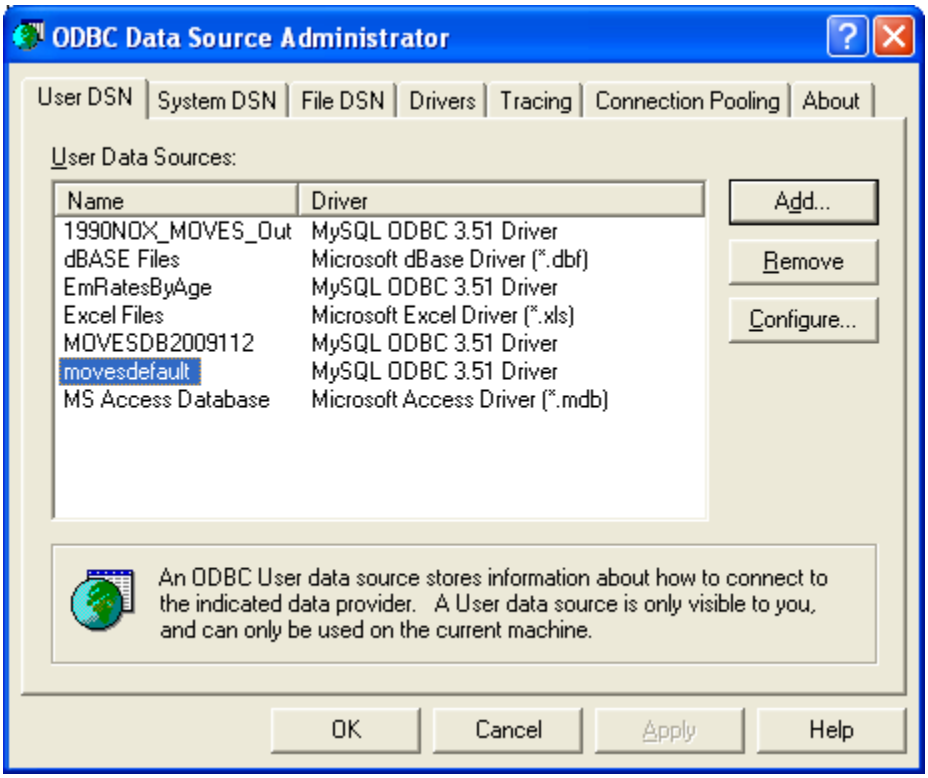


e. Next, the database links need to be defined. The values shown below are for a database called "movesdefault". The database name should be the name of the folder which contains the desired MySQL database files (i.e., the MySQL database name).



f. Once the database name and the data source name list an existing MySQL database name, the database connection can be checked by clicking on the "*Test Data Source*" button.

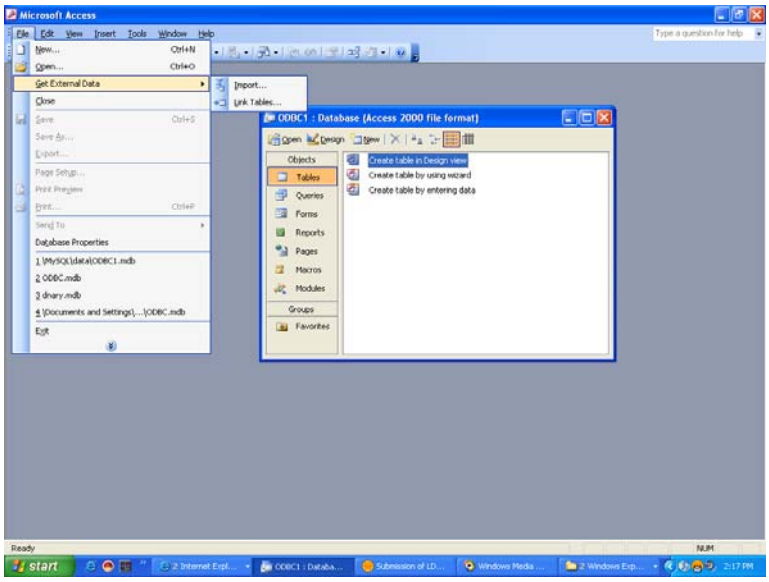
g. The added data source will show up in the list of data sources in the ODBC Data Source Administrator dialog. This will finish the process of creating a new ODBC data source associated with a particular MySQL database.



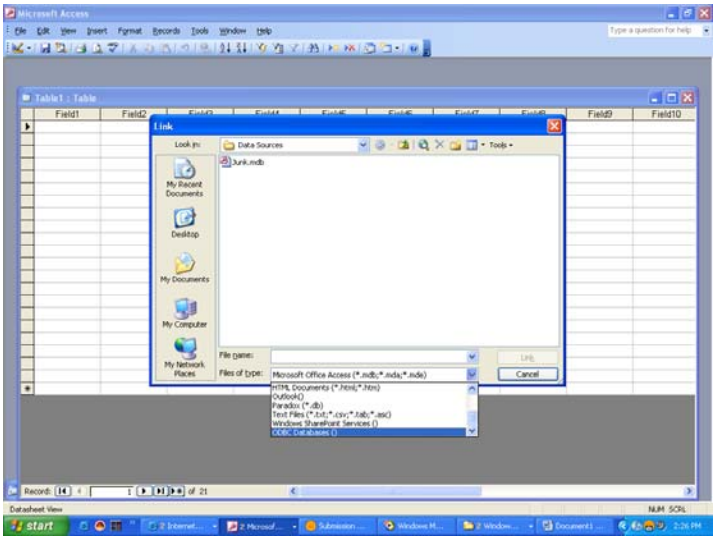
Using Microsoft Access with MySQL Tables

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

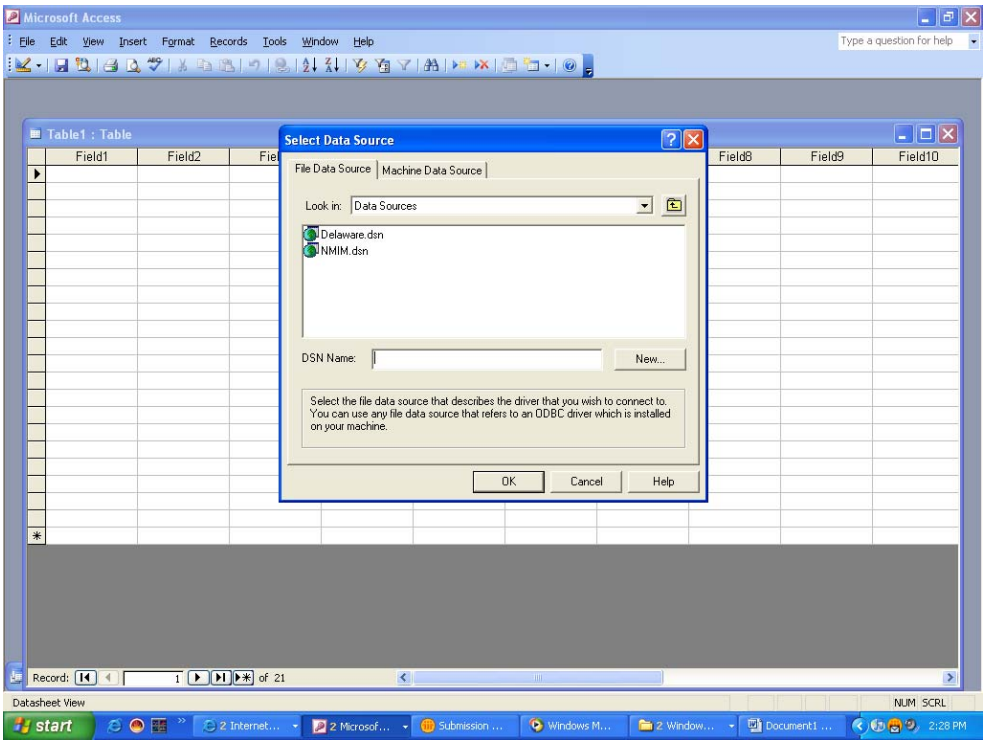
2. On the top menu bar, look for menu items "**File**", "**Get External Data**", then "**Link Tables …**" (see picture below). Click on the item "**Link Tables…**"



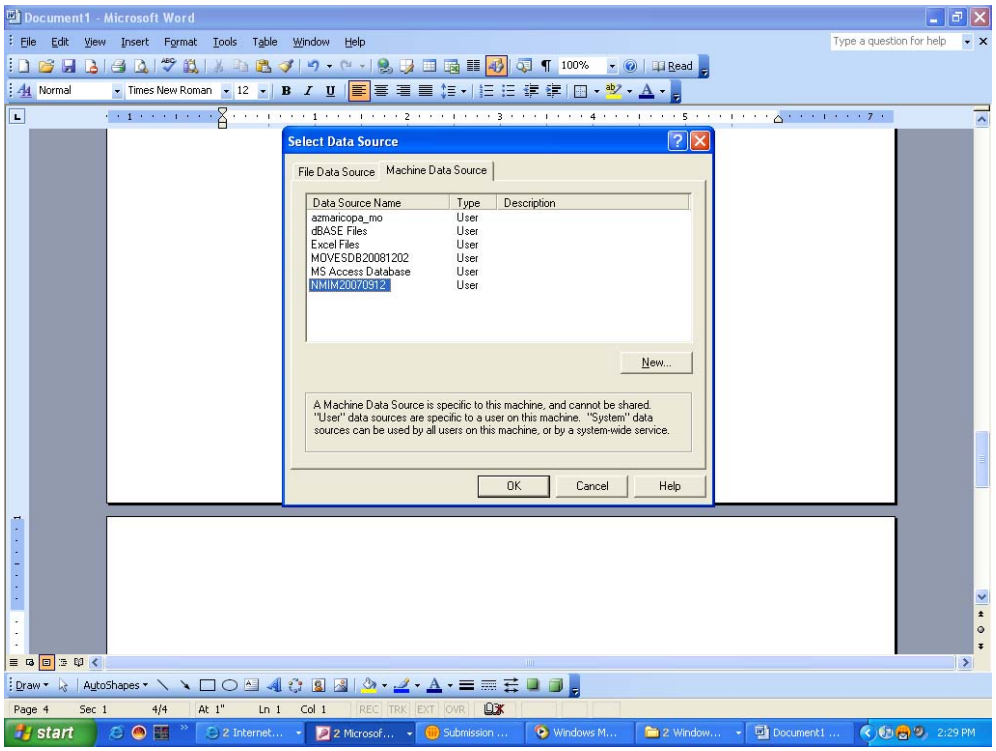
3. On the pop-up window like the one shown below, choose "**ODBC Databases()**" from the drop-down list which is located next to the label "**Files of type**".



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.

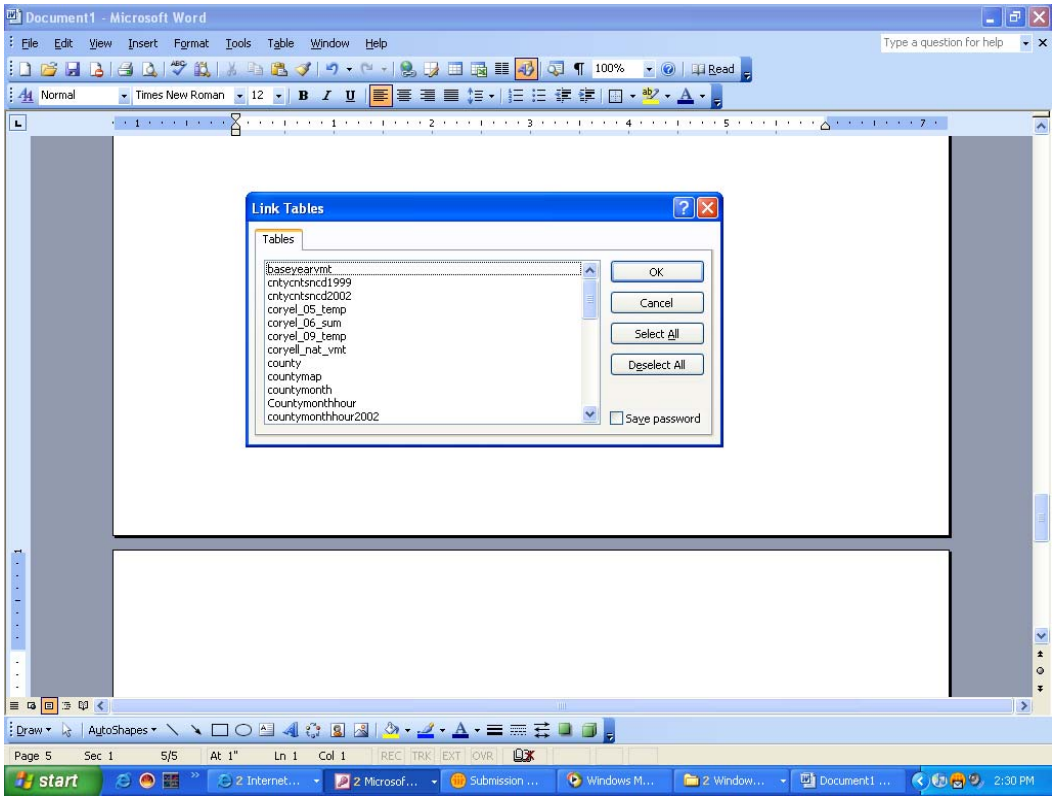


In this example NMIM20070812 is chosen from the **Data Source Name** column. Select **OK** to load the data into Access.



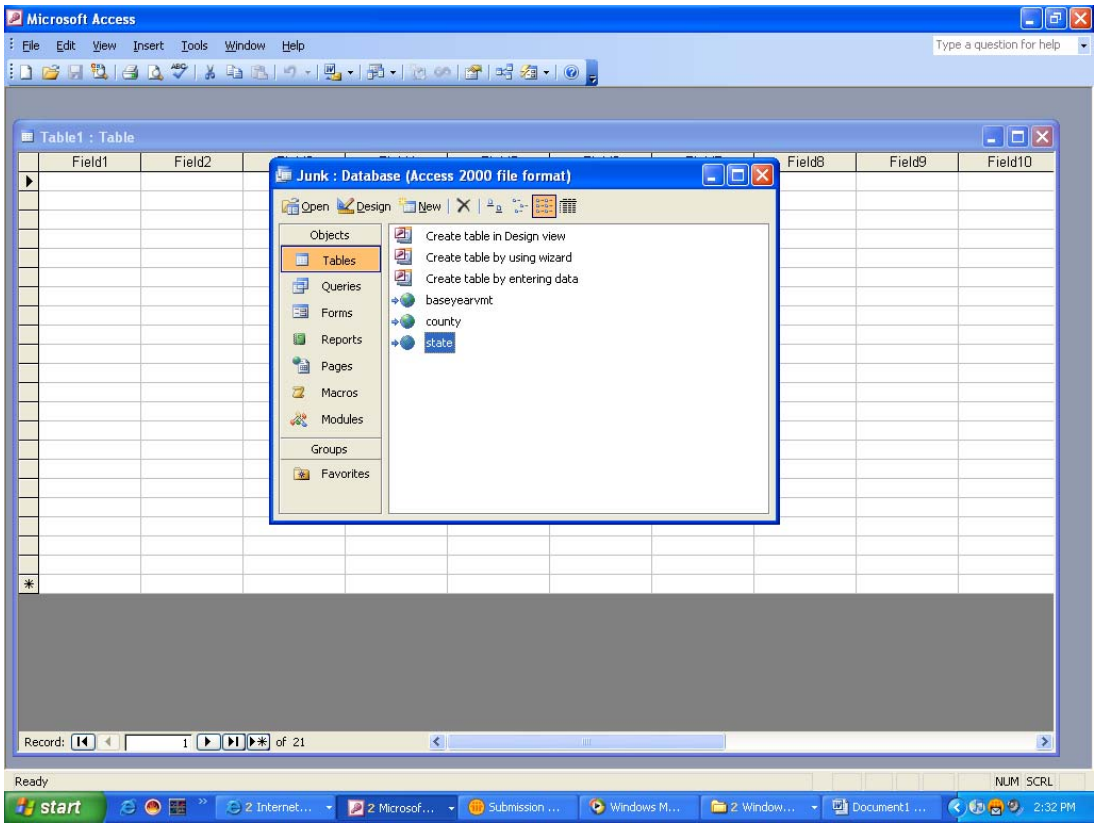
If the **Data Source Name** of the file is not listed, the user must follow the instructions provided in the section, "How to create a DSN Connection to a MySQL database" presented later in this appendix.

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.

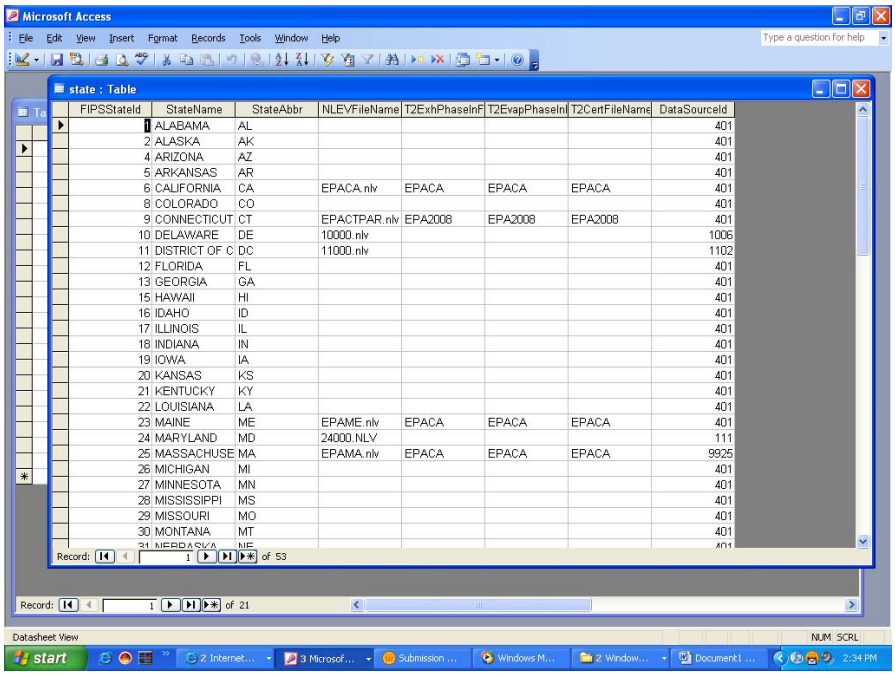


In the screenshot shown below, the user has chosen three tables to work with in MS Access: baseyearvmt, county, and state. The user can now work with these tables in MS Access.

Caution! Edits made to the tables or data in MS Access will not affect the original MySQL tables and cannot be used directly in MOVES.

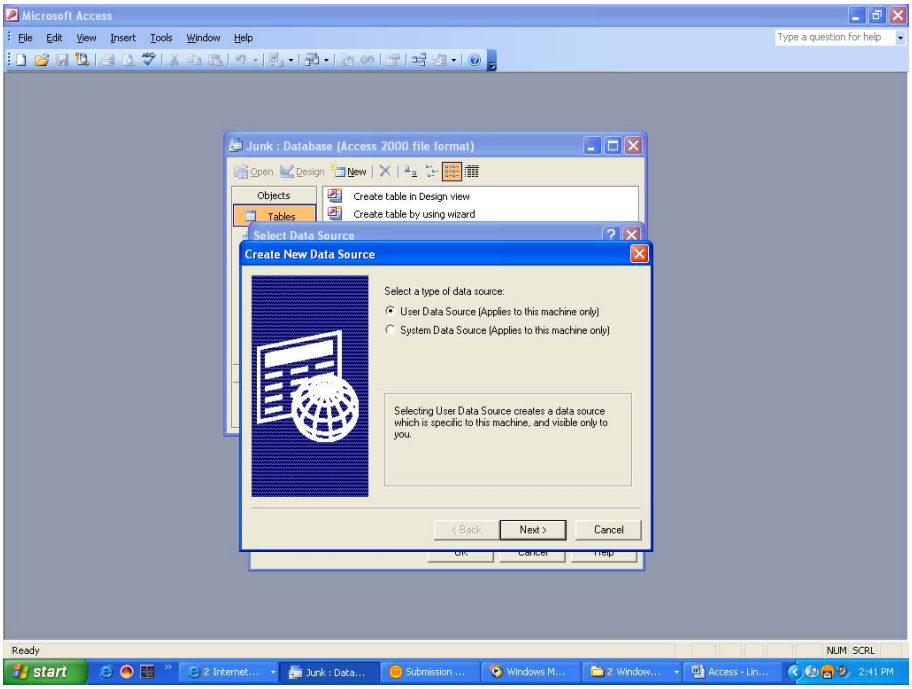

Microsoft Access
File Edit View Insert Tools Window Help
Type a question for help
Table1 : Table
Field1
Field2
Field8
Field9
Field10
Junk : Database (Access 2000 file format)
Open Design New
Objects
Tables
Queries
Forms
Reports
Pages
Macros
Modules
Groups
Favorites
Create table in Design view
Create table by using wizard
Create table by entering data
baseyearvmt
county
state
Record: 1 of 21
Ready
NUM SCRL
start
2 Internet...
2 Microsof...
Submission ...
Windows M...
2 Window...
Document1 ...
2:32 PM

The user can view data specific to a particular state by clicking on the StateName in the table.

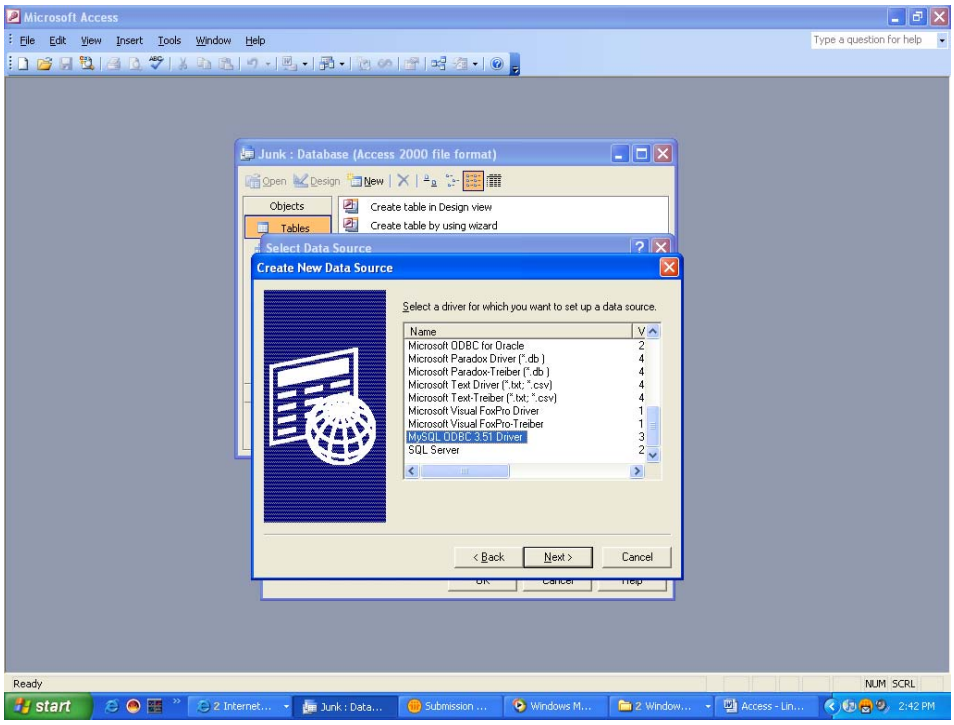


How to create a DSN Connection to a MySQL database.

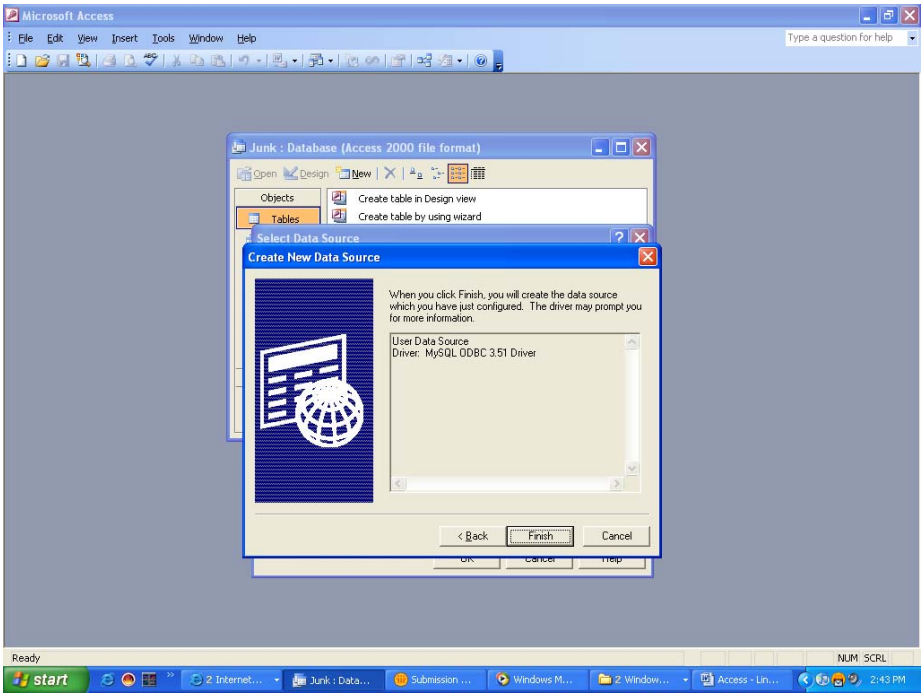
If the user does not find the MySql database referenced in the **Data Source Name** listing in MS Access, a DSN connection must be established. In order to do so, click on the **Machine Data Source** tab from the **Select Data Source** screen. Click the "**New**" button, and select "**Next>**".



At the Create New Data Source Dialog box, select MySQL ODBC 3.51 Driver. Click "**Next >**".

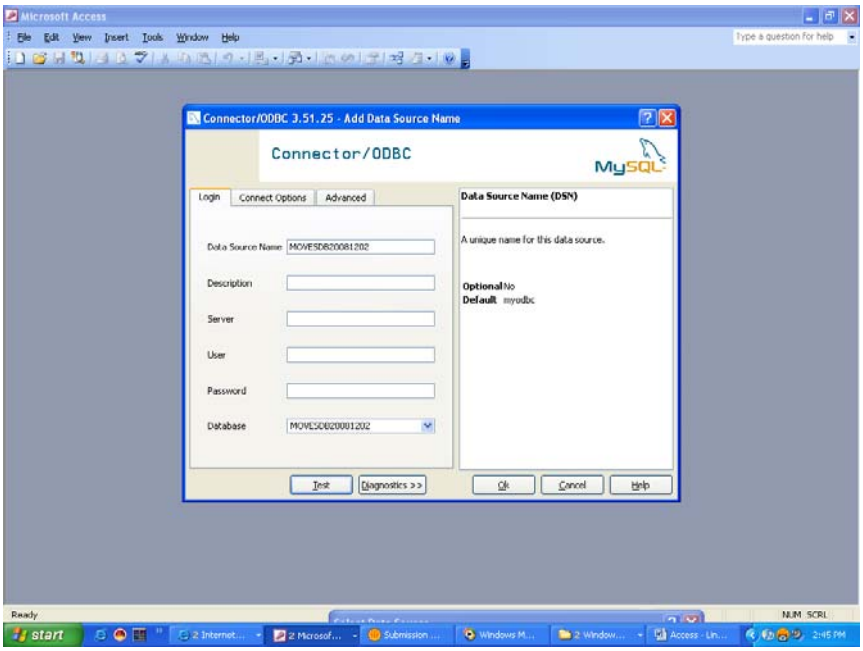


Click "**Finish**" to begin to establish the connection.

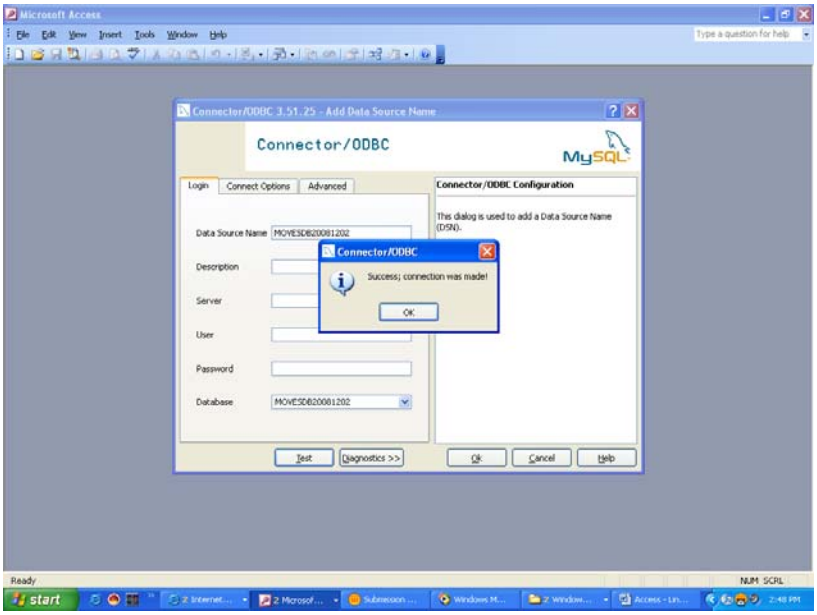


The **Connector/ODBC** screen is used to establish the connection from MS Access to the MySQL database. Use this screen to define the "**Data Source Name**".

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.

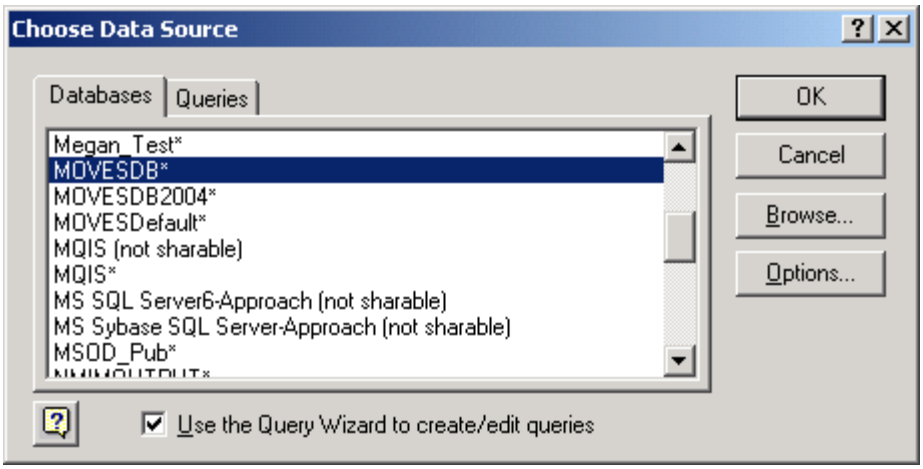
Using Microsoft Excel with MySQL Tables

Note MYSQL Query Browser can also be used to export MYSQL database tables to Excel.

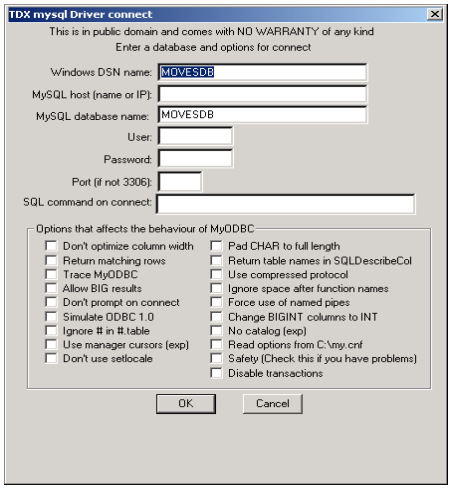
1. Follow the instructions in the section "Adding a MySQL database as an ODBC User Data Source" for adding the desired MySQL database as an ODBC data source.

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 when another computer program executes 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 commands to execute the MOVES Command Line Interface are:

java MOVESCommandLine -r *runspecfile*

[OR]

java -XmxNNNm MOVESCommandLine -r *runspecfile*

[OR]

java MOVESCommandLine -rl *runspeclistfile*

[OR]

java XmxNNNm MOVESCommandLine -rl *runspeclistfile*

where

-XmxNNNm is an optional parameter and NNN is a three digit integer number. For example, –Xmx200m specifies you are asking java to allocate 200 megabytes of heap memory for your runs. This optional parameter is required only if your MOVES runs need more heap memory than the default determined automatically by java and your machine. The actual maximum amount of heap memory that you can ask for depends on your machine. You can increase or decrease the amount of heap memory as your machine allows. For example,

java -Xmx300m MOVESCommandLine -rl *runspeclistfile*

or

java –Xmx100m MOVESCommandLine -r *runspecfile*

"*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. When doing a batch run, you can identify the runspec list text file in the command line using quotes because it is located in a folder with spaces (e.g. “C:\My Documents". However, the command line doesn't work if you put quotes in the runspec list file itself. The command line will think the file doesn't exist.

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:\Program Files\MOVESyyyymmdd, then MOVESCommandLine.class is located at C:\Program Files\MOVESyyyymmdd\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:\Program Files\MOVESyyyymmdd, 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 2.3.3.3), 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
 -i importAllFromTemplates.xml
```

Be sure to place all of this on one line. The "-i" option directs MOVES to the XML file that describes the import actions to occur. Note that this XML file contains a summary of the RunSpec within it, thus allowing wildcards even when importing via the command line. For MOVES installations on Windows 2000 platforms, there may be a problem with the DOS command string length not allowing more than 126 characters. To resolve this, move MOVES to

the root of C: drive (C:\MOVES\), so that the length of the command line string will be shorter. Note that Windows2000 is not a recommended nor supported environment in which to run MOVES.

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 MOVES2010 the user may 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 MOVES2010, 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 MOVES2010, 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 MOVES2010, 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 MOVES2010, 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. See also EPA's guidance on quantifying benefits from retrofits at http://www.epa.gov/otaq/stateresources/transconf/policy.htm

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.

NOTE MOVES database names must be consistent with MYSQL naming conventions. In general, this means they may contain letters, numbers, "_"(underscore) and "$" characters. They may not contain spaces, special characters, or be "reserved words" (SQL words such as "select", "where", "delete", etc.).

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 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 as the import files associated with that RunSpec. 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 first look in the location of the RunSpec when browsing for import files, so keeping them all in one place keeps you from searching for the files that will be imported.	My Documents\MOVES\ [CountyYear]
Decide on the Scale and Calculation Type you wish to use before making selections on any other panels.	Changing the Scale or Calculation Type option after making choices in other panels may require you to re-enter data for 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.	2010_01_01; John Doe; Washtenaw County; 2010; July
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. File location can be added as the "Description" when the file is imported.	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.	
Always "GROUP BY" MOVESRunID, PollutantID, and DayID when summing output results.	If a table includes output from multiple runs, it is very easy to mistakenly sum them together, thereby drastically increasing emissions. Emissions from different pollutants should never be summed. The types of day must be correctly weighted before they can be summed.	SELECT MOVESRunid, DayID, Pollutantid, sum(emissionquant) FROM movesoutput GROUP BY MovesRunid, DayID, Pollutantid;

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

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

In this example run specification (AVFT_Example_1_runspec.mrs), the EPA's Motor Vehicle Emission Simulator (MOVES) was 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 the stringency of car and truck CO_2 standards, while the baseline scenario is similar to the MOVES defaults. To model the policy scenario, new MOVES inputs were created 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. As explained below, the same AVFT file was also 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.).

(A) AVFT files were used to define a new baseline diesel percentage which is higher than that shown in the MOVES 2010 default database.

(B) 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 have tailpipe CO_2 emissions, shifting vehicles to the EV fleet has the same effect as reducing the tailpipe CO2 emissions from the overall fleet. It should be noted this approach is not predicting that EVs will be used to meet tighter CO_2 limits, it is simply the method used here to achieve the desired overall fleet CO_2 levels.

Use of the AVFT:

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. The basis of the MOVES CO_2 estimates is energy consumed and the carbon content of various fuels. By switching a certain portion of vehicles to a fuel with zero tailpipe emissions, we reduce the overall fleetwide tailpipe CO_2 emissions.

Practically, we can model new potential CO_2 limits in MOVES by modifying fuel consumption. Because tailpipe CO2 emissions are directly proportional to liquid fuel consumption, our discussion focuses on miles per gallon (MPG).

Table **F-1** shows the fuel economy assumptions for cars in the baseline and policy scenarios. These fuel economy values are unadjusted values (i.e. the type of values used in Corporate Average Fuel Economy (CAFE) standards), so they are higher than the actual onroad fuel economy. Real world conditions create a gap between on-road fuel economy (calculated by MOVES) and the CAFE values which were used to generate the percent reductions in fuel consumption.

Table F-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
2015	30.7	33.5
2016	30.7	34.8
2017	30.7	36.2
2018	30.7	37.6
2019	30.7	39.1
2020+	30.7	40.7

For simplicity, the baseline and policy scenario assumes equivalent diesel penetration, as shown in Table F-2. The penetration set in the current example differs from that used in the default MOVES database, and is set by the new AVFT.

Table F-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 vehicles of each fuel type 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 F-2, as follows:

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

For example, in the MY 2014 cars 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%

Analysis of Results

The following instructions assume that your results are created in the default database name of CO2_test_database, and that there is only a single run residing in this database. For simplicity, the analysis focuses upon model year 2014.

1. You can check to see that the electric vehicles produce no CO2.

SELECT * FROM CO2_test_database.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 for fuel type 9 emissions equal 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 CO2_test_database.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 CO2_test_database.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 for 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 simple intentionally. It does not 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 single 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 G1 Basic Schematic of the Project

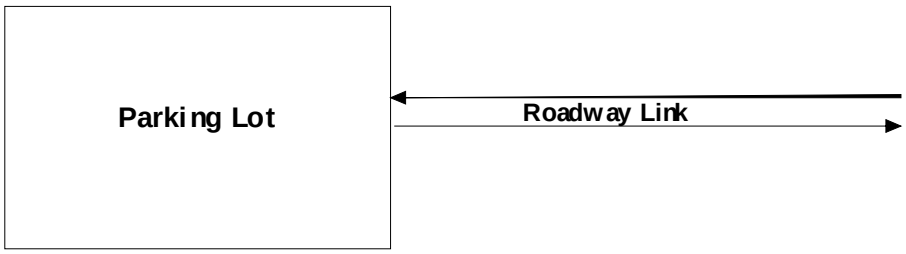


Figure G1 shows a basic schematic of the Project to be modeled. It consists of a parking lot and a two-way road link leading in and out of the parking lot. The specific modeling parameters are shown and discussed in Tables G1, G2 and G3 below.

Table G1
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	23.7 F
Humidity	74.5 %
Road type of links (link1 and link 2)	Rural Unrestricted Access – can represent any non freeway rural road
Roadway Link Length (both links)	0.56 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)	33.6 miles per hour
Link Average Speed – Out (link 2)	31.6 miles per hour
Number of Vehicle in Parking Lot (link 3)	200 vehicle on average

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 custom domain (defined by the user) may be chosen. Also, 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 activity, weekend activity or both. In this example only weekday was run. A temperature in degrees Fahrenheit (23.7 F), and relative humidity of (74.5 %) were also specified. These are the default MOVES values for this county, month and hour. However, any appropriate value may be entered for these parameters.

More than one pollutant / process combination may be modeled in a given project level run if the required inputs are entered. In MOVES, some pollutants are 'chained' to other pollutants. This means that they require the prerequisite calculation of another pollutant(s). If the pollutant to be modeled in the Project level is a 'chained' pollutant, then a complete set of Project level inputs must be entered for all of the required prerequisite pollutants.

Currently, MOVES Project level cannot model evaporative emission processes. However, this capability will be added to future model upgrades.

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. Any of the four roadtypes may be chosen to represent each Project link. The user must also specify a link length in miles for each roadway link they wish to model. In this case because the example is modeling the same roadway in and out of the parking lot, both links are 0.56 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 or all of the MOVES vehicle source types may be included at the same time in a project level run.

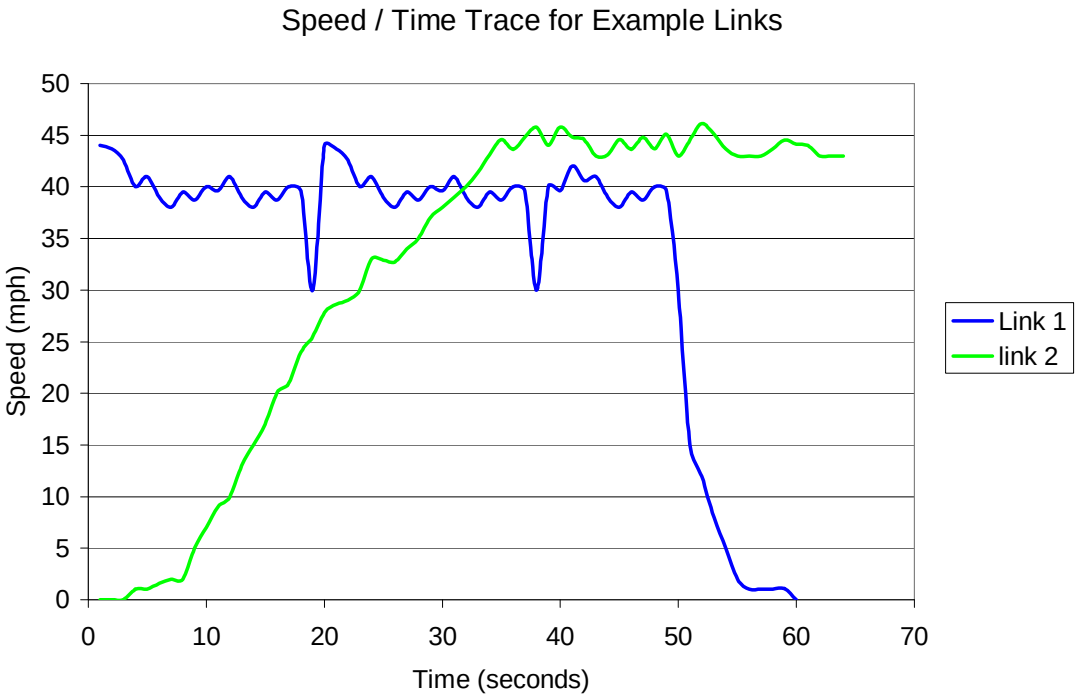
The average speed on each link must also be specified. In this example where drive schedule inputs are used to represent activity of all of the roadway links, the average speed inputs in the Links spreadsheet / input GUI tab are used only for minor internal calculations in the project level model. However, they should match the overall average speed(s) of the individual drive cycles as submitted in the LinkDriveSchedule tab. It is the responsibility of the user to insure that they match – no automatic checks are done.

The average road grade (in percent grade units) may also be specified for each link. This input represents the overall average grade of the entire link not one specific section of it. It is used only if a drive schedule input is not provided (i.e., it will NOT be used in this example). However, in this example a value of zero was entered as a placeholder.

In this example, MOVES will use second-by-second driving schedules to model vehicle operation. If drive schedules are not provided, MOVES uses the average speed and average grade inputs plus default MOVES driving cycles to model the driving behavior. When drive schedules are entered, different emission results may be produced even if the average speed of the driving schedules match the average speed input entered in the Links spreadsheet tab. See Appendix G Section 5.0 for an example.

Figure G2 shows the two driving schedules of this example project in graphical form. Link 1 shown in blue, has driving starting at around 44 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 44 mph cruise. The variable nature of the speed – time curves are indicative of average operation of a number of vehicles / drivers on a road. The driving schedule data and the chart are contained in the input workbook "ProjectExample1_input.xls" in the worksheets "LinkDriveSchedule and LinkDriveSchedule Example Chart."

Figure G2



The distribution of the traffic by MOVES source type is an additional input. It is entered as the SourceTypeHourFraction in the LinkSourceType worksheet, and the LinkSourceType input tab. In this example, because only heavy-duty long-haul trucks are to be modeled, all of the values for SourceTypeHourFraction are set to unity.

For this example, the off-network parameters are shown in Table G2. These four parameters include a vehicle population of 200 vehicles in the parking lot on average during the hour period. This input is an average value over the one hour time period, 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 were started during the hour. If all of the vehicles in the lot on average are started, then this value is 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. *Currently, the parked vehicle input is not active in any of the calculations, but a valid entry must be provided anyway.*

Table G2
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, Table G3 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 G2.

Table G3
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 G4 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 G4 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

A MOVES Runspec **(called ProjectExample1.mrs)** was created to model this example MOVES Project level analysis. For this example, the user should load the Runspec - *Project Example1.mrs*.

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, two roadtypes, one weekday/weekend combination and three pollutant / process combinations. The runspec is provided as an example and the user should use it to become more familiar with this example.

The Project level importer is accessed by either selecting "Project Domain Manager" from the "PreProcessing" menu item at the top of the GUI, or pressing the "Enter/Edit Data" button on the "Geographic Bounds" panel. After one of these selections is made, the Project level importer screen will open.

Caution! In general the user should complete ALL of the runspec entries (i.e., Scale, Time, Geographic Bounds, etc) BEFORE accessing the Project Domain Manager.

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 "ProjectExample1_input" in this example.

The user loads the data into the Project Level database (i.e.,ProjectExample1_input) 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, and the GUI tab should turn from red to green if the data import was successful. This process is repeated for each of the tabs until no more red tabs are present. Various diagnostic messages are provided if the import was not successful. The data should be "Browsed" and "Imported" from the Excel workbook *ProjectExample1_input.xls*.

The inputs are in the spreadsheets.

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

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 **("ProjectExample1_input")** from a list of databases. If the new database is not visible, the user should press the *Refresh* button.

Both the Excel input file **ProjectExample1.xls** and the MOVES project level input database **ProjectExample1_input** are provided in the MOVES package as an assistance to the user. If the user "Browses" and "Imports" their data from the spreadsheet **ProjectExample1.xls,** the user's final database should be the same as **ProjectExample1_input**.

In this example, all of the inputs for the Project Level were read from a single Excel workbook. This was done to make the process easier for the first time user, and to facilitate and simple "Browse" and "Import" data entry. However, if the user is starting from scratch with their own project, it is recommended that they use the *Create Template* button or the *Export Default* button for each of the Project Level input tabs. This will create a set of Excel workbook templates or default data spreadsheets (with a set of individual worksheets - one for each input tab). The user should populate or modify these templates or data tables 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

After using the Project level importer, the user should also notice 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.

Alternatively, neither a DriveScheduleSecondLink or the OpModeDistribution table need be provided. The user may simply specify an average speed for each link in the "Links" table. If this option is utilized, MOVES selects two default driving schedules and uses the average speed input to interpolate between the two cycles to create a new default cycle for that average speed. (See Appendix G Section 4 for more details).

3. MOVES Results for the Example

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

Table G5
MOVES Project Level Example Simulation Results

LinkID	Link Description	NOx emissions (grams)
1	Inbound Road	793.31
2	Outbound Road	120.97
3	Parking Lot (extended idle)	9768.33
3	Parking Lot (start)	4.17

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 results 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 **average** 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.

4. MOVES Project Example #2

This example (the inputs are in Excel Spreadsheet ProjectExample2_input.xls) has virtually all of the same inputs as Project Example #1 discussed in Sections 1 through 4. As a result none of the data and explanations will be repeated. The only difference between the two examples is the ACTIVE use of the Average Speed input in this example instead of the DriveSchedule inputs. The purpose of this brief example is to show that different results for roadway links are obtained from the two different methods of modeling vehicle operation.

In this example, the average speed input of 33.6 mph is imported and used for Link 1 and 31.6 mph is used for Link 2. These inputs are in the Links tab. The average speed / average grade inputs become ACTIVE in MOVES when the specific drive schedule inputs shown in Figure 1 are **NOT** imported / used by the Driveschedule Importer GUI. If the drive schedule data (or operating mode distribution data for roadway links) are imported, they automatically over-ride the average speed inputs in the Links tab.

Instead of user supplied drive schedules being used to represent vehicle operation, built-in default drive schedules are used when average speed inputs are entered. The two input types (drive schedule and average speed) are not necessarily equivalent. The difference can be quantified by comparing the results in Table G6 with those of Table G5.

Table G6
MOVES Project Level Example Simulation Results

LinkID	Link Description	NOx emissions (grams)
1	Inbound Road	731.77
2	Outbound Road	79.45
3	Parking Lot (extended idle)	9768.33
3	Parking Lot (start)	4.17

5. MOVES Project Example #3

This example briefly presents and discussed several real world ramp cycles developed under contractor for EPA. These are provided because the Project Level does not contain default data for modeling freeway ramps. Entering an average speed / grade in the Project Level ***Links*** importer will not model a freeway ramp. The only way a user can model a freeway ramp using the Project Level is to enter a ramp drive schedule in the ***DriveSchedule*** importer or a roadtype operating mode distribution using the OpmodeDistribution input tab.

The attached Excel spreadsheet ***Ramp_Driving_Cycles.xls*** contain several real world drive schedules for various ramp configurations. They are based on a 2004 Sacramento, California driving study. These schedules do not model any actual real world project, but serve as realistic examples of ramp operation. They also function as a starting point for the user to develop actual cycles that precisely fit the needs of their specific project. They can be used in MOVES Project (or modified for use) by defining a new ramp LinkID (i.e., Ramp1, Ramp2, etc) and populating the DriveSchedule importer with a speed-grade / time trace based on ramp driving operation.

The ***Summary*** table in the ***Ramp_Driving_Cycles.xls*** spreadsheet lists the ramp configuration type, the grade change toggle and the ramp metering designation. The table includes only “on-ramps”. No data are available for “off-ramps”. Ramps may be configured in several ways with different operating behavior occurring on each. The most prominent types are the “Diamond” configuration or the “Loop” or “Cloverloop” configuration. The “Diamond” configuration typically has higher speed and acceleration operation than the “Loop” configuration. A ramp may also be “metered” by a traffic light at the end of the ramp that controls access to the freeway. The final ramp configuration is the “Transition” ramp. It occurs when one freeway transitions to another freeway. The individual cycles are also denoted by the general grade change of the ramp. Three possibilities are provided for the “Diamond” and “Loop” configurations.

Appendix H - Scenario 3: County Data Manager Example and Basic MySQL Queries for Analyzing Output

1. Explanation of Scenario 3

The County Data Manager is expected to be used extensively by users, particularly by nonattainment or maintenance areas when conducting SIP or conformity analyses. This tool provides the user the ability to import and edit local data, rather than using MOVES defaults. This appendix will also go through the MOVESOutput table in the MySQL Query Browser to give users some basic queries that can be used for their own analyses.

In this scenario, the County Data Manager is 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 Data Manager is 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 will reflect that assumption. On the **Pollutants and Processes** panel, all processes for Volatile Organic Carbons (VOCs) and Oxides of Nitrogen (NO_X) are selected because these pollutants are associated with the formation of ground-level ozone. If VOCs are selected, several other pollutants and processes must be selected as well because they are needed when calculating VOC emissions. Therefore, this run contains a fair number of pollutant-processes and will take a fair amount of time to run (about 16 minutes on a computer with a dual-core processor), but the variety of the output will be useful later when analyzing the output in the MySQL Query Browser. If the user simply wants practice with the process of using the County Data Manager and importing files, it is recommended that the user select only NO_X processes on the **Pollutants and Processes** panel as evaporative emissions of hydrocarbons take the most time in MOVES.

The RunSpec should be filled out in its entirety before the County Data Manager is used. In this example, selections were limited to reduce the run-time of the RunSpec, but some variation is included to emphasize how output can be post-processed in MySQL.

Open the file name "examplecity_2013_july.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_2013_july_out). Then select the RunSpec **Geographic Bounds** panel and notice that Custom Domains require the user provide additional information about the area. Users can change any of these fields (except County ID because the input files have countyID=99001), but should be aware that doing so will result in different emission results. Next, click the "Enter/Edit Data" button to 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_2013_july_in).

Caution! If any changes are made to the RunSpec after the database has been created, the database may no longer be correct for the revised RunSpec. This is because some "core" tables are populated based on the selections in the RunSpec at the time of the creation of the database and these core tables can only be changed directly in MySQL (i.e., there are no tabs to edit these core tables). Users should take care in if changing the RunSpec while using an existing database and should always double-check the tab headers to ensure all the necessary data has been provided after making a change to the RunSpec.

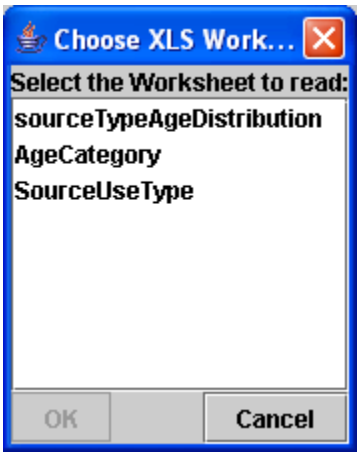
Next, notice that the Fuel Formulation, Ramp Fraction, Zone Road Activity, and I/M Programs tabs all start with green checks for different reasons. For Fuel Formulation, as long as default fuel formulations are referenced in the Fuel Supply tab, then no data has to be imported; however, if a new fuel formulation is created or if the properties of an existing fuel formulation are changed, then the new fuel formulations should be imported. For Ramp Fraction, the default fraction of 8% of VHT is applied if no data is imported on this tab, but users can import new values if desired. The Zone Road Activity tab erroneously shows a green check, but data MUST be imported for MOVES to calculate results properly; a value of 1 should be entered for the SHOAllocFactor for each road type so that all of the VMT input by the user is assigned to the custom domain. Finally, the I/M Programs tab starts with a green check because there are no default programs in the custom domain that have to be accounted for. Of these 4 tabs, data will be imported for Zone Road Activity and I/M Programs.

For all other tabs, files containing the area-specific data must be imported. 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_2013_july_[TABNAME]

Users can open the files before importing them to view how the data is formatted and can refer to the supporting worksheets for clarification on the numerical codes or abbreviations. Notice that for some files, wildcards are used to greatly reduce the number of rows that have to be entered (e.g. Average Speed Distribution - 64 with wildcards vs. 11,520 without).

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_2013_july_agedistribution.xls"). After the file has been selected, a small pop-up window appears:



Select the correct worksheet (in this example, it will generally be the first worksheet listed and will usually have a name similar to the tab name) and click "OK." For the Vehicle Type VMT tab, there are four worksheets within the file "examplecity_2013_july_vehicletypevmt.xls" that should be imported: HPMSVTypeYear, monthVMTFraction, dayVMTFraction, and hourVMTFraction. The names of the worksheets correspond to the names of the database tables for which data is being provided.

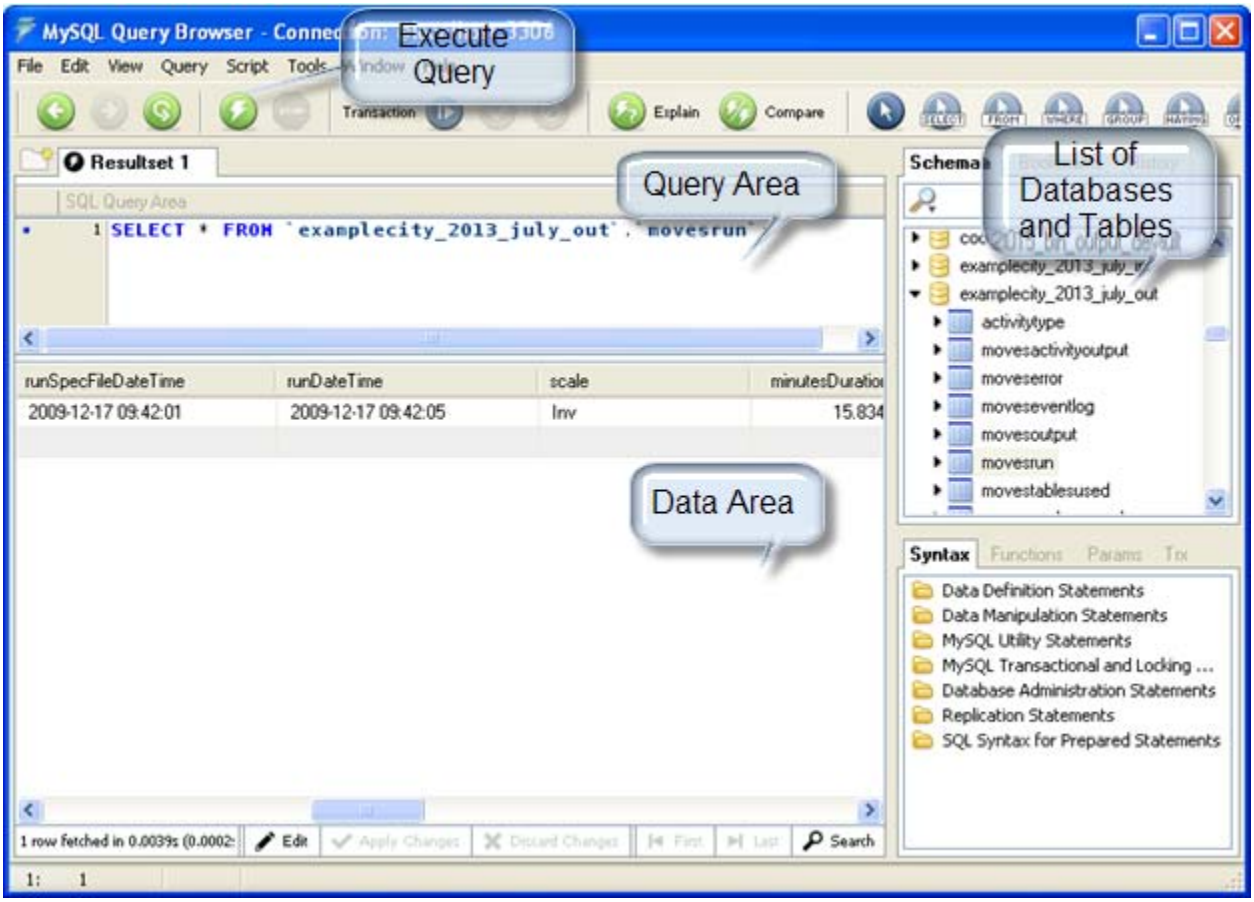
Next click the "Import" button. The "Messages" box should say which table was imported and "Import Complete." If there were problems with the import, error messages here would describe the problem. In case of error, the user should revise the input file, save the changes, and then click the "Clear Imported Data" button before reattempting the import.

The County Domain Manager can be used to input data for a custom domain (as shown in this example) or a specific county. Currently, the only difference is that the custom domain option includes a "Zone Road Activity" tab. 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 MOVES2010 and values of 1 should be entered for all road types within the zone.

Once all the files have been imported, the database tables have been populated. Click "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. 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.

4. Analyzing the Results in the MySQL Query Browser

First, a brief summary of the major pieces of the MySQL Query Browser.



The top left box is the Query Area, bottom left is the Data Area, and top right is the List of Databases/Tables. The tables within a database can be revealed by clicking the black triangle just to the left of the database name. There are queries below that have the following syntax:

SELECT * FROM `examplecity_2013_july_out`.`[table]`;

These queries can be auto-generated by dragging the table name from the right-hand List of Tables and dropping it in the Query Area or Data Area. If the table name is dropped in the Query Area, the text for the query is generated, but the query it self is not executed; so the user must either click the Execute button (lightning bolt) or press CTRL + Enter. However, if the table name is dropped in the Data Area, the query will be generated in the Query Area and auto-executed with results immediately appearing in the Data Area.

Now that some of the basics have been covered, the results from the example can be analyzed. Listed below are MySQL Queries and some discussion of the results they generate. Users should be able to copy and paste the queries into the Query Area of MySQL as long as the output database name is "examplecity_2013_july_out"; if the user provided a different output database name, that name should be used wherever "examplecity_2013_july_out" appears.

1) SELECT * FROM `examplecity_2013_july_out`.`moveserror`;

Users can look at the MOVESError log to make sure the run executed normally. There should be no results from this query.

2) SELECT * FROM `examplecity_2013_july_out`.`movesrun`;

The MOVESRun table gives some basic information about the RunSpec, the description, the date and length of time, etc. If there are multiple runs in the database, there will be multiple MOVESRunID entries here.

3) SELECT * FROM `examplecity_2013_july_out`.`movesactivityoutput`;

The MOVESActivityOutput table contains all the activity results from the RunSpec and should have 1446 rows (there is a bar below the data area that tells you how many rows a query returns). In the RunSpec, the distance traveled and population options were selected. Both of these values were inputs, so let's make sure the input values were carried through to the output correctly. Population is easier, so we'll look at that first.

4)
```
SELECT sourcetypeID, SUM(activity) AS population
FROM `examplecity_2013_july_out`.`movesactivityoutput`
WHERE activitytypeID=6
GROUP BY sourcetypeid
ORDER BY sourcetypeid;
```

You'll notice the query has gotten more complicated, but it can be easily explained. The first line is the SELECT statement, which states the fields from the table that will be displayed. Since population is not time dependent and we're only looking at one county, the only important field is the sourcetypeID itself; however, a sum of the activity is required because output was differentiated by fuel type, but that is not of interest right now. The SUM(activity) will be renamed in the resulting table "AS population". The second line is the FROM statement, which just identifies the database and table that should be used. The third line is the WHERE clause, which specifies that only activitytypeID=6 should be used when querying the table (since we're just looking at population). The fourth line is the GROUP BY clause, which is used when a sum is calculated. Since the clause says, "GROUP BY sourcetypeid", all data of different source types will be kept separated, but any data within the same source type will be summed (so the different populations for each fuel type will be added together). The fifth row is the ORDER BY clause, which simply states the order in which the resulting information will appear.

The resulting table from this query should have just 3 rows and the populations should be (after rounding) 500,000; 300,000; and 50,000 for source types 21, 31 and 32, respectively. These values match the populations that were imported.

5)
```
SELECT dayid,sourcetypeid, SUM(activity)*dayID*31/7 AS PortionofMonthVMT
FROM `examplecity_2013_july_out`.`movesactivityoutput`
WHERE sourcetypeid=21 AND activitytypeid=1
GROUP BY dayid
ORDER BY dayid;
```

This statement is the beginning to calculate the VMT for passenger cars (sourcetypeid=21), but doing so is not a straight-forward process and that fact can be seen in the SUM statement above (notice that the GROUP BY clause only has dayid because the WHERE clause identifies that

only 1 sourcetypeid will be queried, so grouping by sourcetypeid is not necessary, but would not affect the query if it were included). The temporal output for the RunSpec is hour, so that means the VMT output was originally for one hour of one of the types of days; therefore, the VMT should be summed within the dayID to get the total for that type of day. Then, this query adds two additional steps: 1) by multiplying by the dayid, the product is the VMT on that type of day in a week; and 2) by multiplying by 31/7 (the number of days in July divided by the number of days in the days in a week), which is the number of weeks in July, this second product gives the VMT on that type of day in the entire month.

The resulting table should have just 2 rows as can be seen in the table below. An additional column has been added using an Excel spreadsheet to sum the VMT from the two types of days.

dayid	sourcetypeid	PortionofMonthVMT	MonthVMT
2	21	1.21E+08	
5	21	3.44E+08	4.65E+08

The monthVMTFraction table assigned a value of 1 to July, so the VMT that was input was essentially a monthly VMT for July and we can see that the MonthVMT is equal to the value that was input for HPMSVTypeID of 20 (examplecity_2013_july_vehicletypevmt.xls, worksheet "HPMSVTypeYear"). There are ways to calculate the MonthVMT entirely within MySQL, but that involves either creating tables or using subqueries, which is beyond the scope of this example. Therefore, once users get to this point, use of an external spreadsheet tool is recommended, unless the user is familiar with more complex MySQL queries.

More detail can be acquired using the query above as a template (e.g., adding "roadtypeID" to the SELECT, GROUP BY, and ORDER BY clauses to see the VMT on each road type to make sure the road type distribution was applied correctly; calculating VMT for HPMSVTypeID=30). Users are encouraged to experiment with the query above to improve their MySQL skills and better understand how to compare inputs with outputs to make sure the results are logical.

6) SELECT * FROM `examplecity_2013_july_out`.`movesoutput`;

This table should contain 23432 rows and gives all the emissions output with a significant amount of detail since output is differentiated by fuel type, emission process, road type, and source type. Therefore, using MySQL to condense the results holds great potential for simplifying the post-processing.

```
7) SELECT       monthid, pollutantid, SUM(emissionquant) AS grams
        FROM `examplecity_2013_july_out`.`movesoutput`
        WHERE dayid=5 AND pollutantid IN (3,87)
        GROUP BY dayid, pollutantid
        ORDER BY pollutantid;
```

This query will greatly simplify the original output table from 23,000 rows to 2, but a major reason for this is that not all the data is being reported.

monthid	pollutantid	grams
7	3	2.11E+07
7	87	1.21E+07

Some assumptions were made to reduce the reported output that may be similar to those users will make in their analyses, such as only a single weekday is being looked at and the only pollutants of interest are NO_X (3) and VOCs (87). Notice that in the WHERE clause that a new command was used, "pollutantid in (3,87)"; the IN command is a more concise way of saying "pollutantid=3 AND pollutantid=87" and can be used with any field that has multiple entries.

Once again, users are encouraged to vary this query to see the results for different conditions (e.g. choose other pollutants; see emission results from each source type or on the different road types; etc.).

Appendix I – Stage II Refueling Control Programs

Stage II refueling emission control programs are intended to reduce HC and associated air toxics emissions by reducing the amount of gasoline vapor that escapes to the atmosphere during refueling. The amount of reduction depends on whether the vehicle has an onboard recovery system and the level of uncontrolled emissions. The uncontrolled emissions are calculated from inputs such as fuel RVP, vehicle fuel economy, and temperature parameters. Stage II programs are run by state, local or tribal governments and the effects of these programs will vary depending on the number of locations and the sales volume of stations equipped with Stage II recovery systems and their average state of repair. See the MOVES Software Design Reference Manual for more information about how MOVES calculates emissions from refueling.

The MOVES database contains information about all of the existing Stage II programs by county based on the parameters used for the 2005 National Emisson Inventory (NEI). For the initial release of MOVES, the effects of Stage II programs can only be altered by users by manually editing the tables used by MOVES to obtain the Stage II control information. Future versions of MOVES are expected to include graphical user interface (GUI) tools to assist in altering Stage II program effects for individual counties. Until these tools are available, this appendix is intended to assist users in making changes to the default parameters related to Stage II program effects.

Stage II refueling emission control programs can only affect refueling losses that occur during refueling. Onboard Refueling Vapor Recovery (ORVR) systems on modern vehicles are designed to minimize the refueling losses without Stage II controls. These reductions are already accounted for by MOVES, so that the additional control of Stage II will only affect the remaining refueling losses from these ORVR vehicles.

County Year Table

The MOVES database contains a table named CountyYear which contains information about each county for every calendar year. This table contains two fields that determine the Stage II program effects:

- refuelingVaporProgramAdjustment
- refuelingSpillProgramAdjustment

The refuelingVaporProgramAdjustment field is a number between zero and one (1.0) which indicates the fraction that is the reduction in full refueling displacement vapor losses that result from the Stage II recovery program that county in that calendar year. The refuelingSpillProgramAdjustment field is a number between zero and one (1.0) which indicates the fraction that is the reduction in full refueling fuel spillage losses that result from the Stage II recovery program that county in that calendar year. A value of zero would indicate that the program had no effect and a value of one would indicate that all vehicle refueling emissions had been eliminated.

Updating Refueling Adjustments Using MySQL Query Browser

The simplest way to change an existing Stage II program effect is to use the MySQL Query Browser application to open the appropriate table and manually change the existing values, but NEVER change values in a default MOVES database provided by EPA. MOVES is designed to easily provide replacement values for default values using MySQL tables via the **Manage Input Data Sets** panel on the MOVES graphical user interface (GUI). Using the **Manage Input Data Sets** panel, type the name you wish to use for the database which will hold your Stage II estimates into the Database drop-down box. Now click on the "Create Database" button and your will create a database to hold your information. The name will have to meet the criteria for database names (no spaces, no special characters) and we suggest it include text to help you identify its contents and purpose (for example, StageII_Input).

Once you have an input database, you can open the MySQL Query Browser to work with the tables. If the Query Browser was already open, you may need to right-click in the Schemata panel and refresh the list of database. Once you find your database, double-click on the name and that will show you all of the tables it contains. The database will contain empty versions of every table used by MOVES. The table that contains the Stage II program adjustments is called CountyYear and contains only four fields:

- countyID
- yearID
- refuelingVaporProgramAdjustment
- refuelingSpillProgramAdjustment

Using the Query Browser Edit function (at the bottom of the result window) you can add or alter the contents of the table. Be sure to execute the Apply Changes button to save any alterations.

Now return to the Manage Input Data Sets panel and select your database from the pull down menu. You can add additional description text and then click the add button. When you include this database in your run specification, MOVES will use your values in preference to any default values in the MOVES database.

The other empty tables may be deleted from your database. Leaving them will not harm your run specification (since the tables are empty), but they can be confusing. To delete empty tables, go to the Query Browser, double click on the database to show the tables and right click on the table you wish to delete. Choose "drop table" from the menu and the table will be removed from that database. Multiple tables can be selected by using the CTRL or Shift buttons.

Appendix J – MOVES "Decoder"

MOVES "Decoder"

SourceType	
sourcetypeid	sourcetypename
11	Motorcycle
21	Passenger Car
31	Passenger Truck
32	Light Commercial Truck
41	Intercity Bus
42	Transit Bus
43	School Bus
51	Refuse Truck
52	Single Unit Short-haul Truck
53	Single Unit Long-haul Truck
54	Motor Home
61	Combination Short-haul Truck
62	Combination Long-haul Truck

Process	
processid	processName
1	Running Exhaust
2	Start Exhaust
9	Brakewear
10	Tirewear
11	Evap Permeation
12	Evap Fuel Vapor Venting
13	Evap Fuel Leaks
15	Crankcase Running Exhaust
16	Crankcase Start Exhaust
17	Crankcase Extended Idle Exhaust
18	Refueling Displacement Vapor Loss
19	Refueling Spillage Loss
90	Extended Idle Exhaust

Pollutant	
pollutantid	pollutantname
1	Total Gaseous Hydrocarbons
2	Carbon Monoxide (CO)
3	Oxides of Nitrogen
5	Methane (CH4)
6	Nitrous Oxide (N2O)
20	Benzene
21	Ethanol
22	MTBE
23	Naphthalene
24	1,3-Butadiene
25	Formaldehyde
26	Acetaldehyde
27	Acrolein
30	Ammonia (NH3)
31	Sulfur Dioxide (SO2)
32	Nitrogen Oxide
33	Nitrogen Dioxide
79	Non-Methane Hydrocarbons
80	Non-Methane Organic Gases
86	Total Organic Gases
87	Volatile Organic Compounds
90	Atmospheric CO2
91	Total Energy Consumption
92	Petroleum Energy Consumption
93	Fossil Fuel Energy Consumption
98	CO2 Equivalent
100	Primary Exhaust PM10 - Total
101	Primary PM10 - Organic Carbon
102	Primary PM10 - Elemental Carbon
105	Primary PM10 - Sulfate Particulate
106	Primary PM10 - Brakewear Particulate
107	Primary PM10 - Tirewear Particulate
110	Primary Exhaust PM2.5 - Total
111	Primary PM2.5 - Organic Carbon
112	Primary PM2.5 - Elemental Carbon
115	Primary PM2.5 - Sulfate Particulate
116	Primary PM2.5 - Brakewear Particulate
117	Primary PM2.5 - Tirewear Particulate

Day	
dayID	dayName
2	Weekend
5	Weekdays

Road Type	
roadtypeid	roaddesc
1	Off-Network
2	Rural Restricted Access
3	Rural Unrestricted Access
4	Urban Restricted Access
5	Urban Unrestricted Access

FuelType	
fuelTypeID	fuelTypeDesc
1	Gasoline
2	Diesel Fuel
3	Compressed Natural Gas
9	Electricity

Activity		
activityTypeID	activityType	activityTypeDesc
1	distance	Distance traveled
2	sourcehours	Source Hours
3	extidle	Extended Idle Hours
4	sho	Source Hours Operating
5	shp	Source Hours Parked
6	population	Population
7	starts	Starts

SCCV Type		
SCCVtypeID	PART5SCCV typeDesc	MOBILE6SCCVtypeDesc
1	LDGV	1, 'LDGV', 'Light Duty Gasoline Vehicles (LDGV)'
2	LDGT1	Light Duty Gasoline Trucks 1 & 2
3	LDGT2	Light Duty Gasoline Trucks 3 and 4
4	HDGV	Heavy Duty Gasoline Vehicles 2B thru 8B and Gasoline Buses
5	MC	Motorcycles (MC)
6	LDDV	Light Duty Diesel Vehicles (LDDV)
7	LDDT	Light Duty Diesel Trucks 1 thru 4 (LDDT)
8	2BHDDV	Heavy Duty Diesel Vehicles (HDDV) Class 2B
9	LHDDV	Heavy Duty Diesel Vehicles (HDDV) Class 3, 4, and 5
10	MHDDV	Heavy Duty Diesel Vehicles (HDDV) Class 6 and 7
11	HHDDV	Heavy Duty Diesel Vehicles (HDDV) Class 8A and 8B
12	BUSES	Heavy Duty Diesel Buses (School and Transit)