

MOVES2004 User Guide

Draft

MOVES2004 User Guide

Draft

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

Prepared by the MOVES Development Team

Ed Glover
John Koupal
Mitch Cumberworth
Dave Brzezinski
Gwo Shyu
Bob Gianelli
Larry Landman
Megan Beardsley
James Warila
Ed Nam
Harvey Michaels
Connie Hart
Sujan Srivastava
Gene Tierney

Table of Contents

1. GETTING STARTED	1
ABOUT MOVES2004	1
ABOUT MOVES2004 DOCUMENTATION	2
ABOUT THIS MANUAL	2
CAUTIONS, NOTES, AND TIPS	4
OTHER DOCUMENTATION AND ONLINE HELP	4
WHAT'S NEXT	5
INSTALLATION	6
SYSTEM REQUIREMENTS	6
TESTING YOUR INSTALLATION	7
2. USING MOVES2004	9
OVERVIEW OF MOVES RUNSPEC USER INTERFACE	9
MAIN MENU BAR	10
NAVIGATION LIST	22
DETAIL PANEL	23
STARTING MOVES	24
DESCRIBING RUNSPEC	26
SELECTING SCALE	27
SELECTING MACROSCALE GEOGRAPHIC BOUNDS	28
NATION	29
STATE	30
COUNTY	31
SELECTING TIME SPANS	33
TIME AGGREGATION LEVEL	34
YEARS	35
MONTHS	36
DAYS	36
HOURS	36
SELECTING VEHICLES/ EQUIPMENT	38
SELECTING ROAD TYPE	39
SELECTING ROAD TYPE	40
SELECTING POLLUTANTS AND PROCESSES	42
PROCESSES	43
POLLUTANTS	44
SPECIFYING ADDITIONAL DATABASES (INPUT DATA SETS) (OPTIONAL)	46
SELECTING STRATEGIES	49
ADDING OBJECTS	49
DELETING OBJECTS	50
EDITING OBJECTS	50
SAVING AND TRANSFERRING OBJECTS	51
VARYING VEHICLE FUELS AND TECHNOLOGIES	53

NORMALIZING DATA	55
ADDING MODEL YEARS	55
SPECIFYING OUTPUT DIMENSIONS	56
SPECIFYING DATABASE, TIME PERIOD AND UNITS IN OUTPUT	56
SPECIFYING EMISSION DISTINCTIONS IN OUTPUT	60
<u>3. EXECUTING EXAMPLE MOVES2004 RUN SPECIFICATION</u>	<u>63</u>
<u>4. CUSTOMIZING MOVES 2004</u>	<u>65</u>
LOCAL CUSTOMIZATION	65
“WHAT-IF” ANALYSIS CUSTOMIZATION	66
FUEL AND VEHICLE TECHNOLOGY PENETRATION	66
ENERGY AND EMISSION PERFORMANCE	67
FUEL PATHWAY OPTIONS	67
ACTIVITY PATTERNS	67
<u>APPENDIX A. USING GREET TO UPDATE THE WELL-TO-PUMP EMISSION FACTORS</u>	<u>69</u>
<u>APPENDIX B. LINKING MYSQL TABLES FROM MS ACCESS (INCLUDES ODBC USAGE)</u>	<u>71</u>

1. Getting Started

MOVES2004 is EPA's initial release of the MOtor Vehicle Emission Simulator. MOVES is intended to include and improve upon the capability of previous modeling tools (i.e., MOBILE and NONROAD) and, eventually, to replace them with a single, comprehensive modeling system that better addresses current mobile source emission analysis needs.

About MOVES2004

MOVES2004 can be used to estimate national inventories and projections at the county-level for energy consumption (total, petroleum-based and fossil-based), Nitrous Oxide (N₂O), and Methane (CH₄) from highway vehicles. It also includes a version of Argonne National Laboratory's GREET model, to include life cycle (i.e., well-to-pump) effects in the estimate of energy consumption and emissions. Future versions of the model are planned to estimate pollutants from additional mobile sources such as aircraft, locomotives, and commercial marine activity, estimate non-highway mobile source emissions, estimate criteria pollutant emissions, and operate at smaller scales.

MOVES2004 is distributed free of charge by the U.S. EPA pursuant to the GNU General Public License (GPL). It is written in Java™ and the MySQL relational database management system, a product of MySQL AB, which MySQL AB also allows to be distributed pursuant to the GPL. Its principal user inputs and outputs, and several of its internal working storage locations, are

MySQL databases. A “default” input database, covering 3222 counties of the United States and which supports model runs for calendar years 1999 - 2050 is included with the model. MOVES2004 interfaces with a version of the GREET model produced by Argonne National Laboratory, which is included on the MOVES2004 installation CD.

MOVES2004 has a “master – worker” program architecture which enables multiple computers to work together on a single model run. A single computer can still be used to execute MOVES2004 runs by installing both the master and worker components on the same computer.

About MOVES2004 Documentation

MOVES2004 documentation extensively covers all aspects of design, data sources and inputs, installation, and operation of MOVES. The following should highlight what is available in this manual, how to best use it, and where additional information may be found.

About this Manual

The MOVE2004 User Guide focuses on operating MOVES software to do a run specification (RunSpec). Chapters focus on the specific functions and inputs the user is asked to provide. This manual assumes background knowledge of MOVES terminology and design concepts. For more explanation about unfamiliar terms or the design of the software, see the next section “Other Documentation and Online Help”.

MOVES2004 is designed to work only in Microsoft Windows 2000 and later environment (i.e., Versions Windows 2000, NT and XP). This manual assumes that the user is familiar with the basics of a Microsoft Windows based interface. These basics include mouse operation, opening and closing files and windows, switching between windows and panels and selecting menu items. If the user is unfamiliar with these basic computer usage topics they should refer to a Microsoft Windows user guide prior to proceeding with this user guide.

It should be noted that some of the graphics in this guide appear to be out of focus or illegible in some cases. This lack of clarity in some graphics was not done intentionally, but is the result of utilizing less than optimal resolution screen shots and the reduction of bitmap images so that they fit on a single portrait page. The authors regret any inconvenience or confusion that this may cause.

Cautions, Notes, and Tips

Throughout this document, certain information is highlighted to make it easier to find solutions to problems or avoiding errors.

- ⚠ CAUTION!** Cautions must be observed to avoid errors in execution or to assure execution will take place as desired.
- 📌 NOTE** Notes contain important information about the panel being described
- !TIP** Tips contain hints for input or better operation of the run.

Other Documentation and Online Help

Additional documentation covering the following topics is available for MOVES2004:

- **Installation:** An Installation Guide is included on the MOVES installation CD. It guides the user through the process of installation and initial execution of the MOVES program.
- **Software Design:** The document "MOVES Software Design Reference Manual" 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
- **Fleet & Activity Inputs:** "MOVES2004 Highway Vehicle Population and Activity Data" explains the data sources and methods used to estimate default vehicle populations and activity used by MOVES2004.
- **Energy & Emission Inputs:** "MOVES2004 Energy and Emissions Inputs" explains the data sources and methods

used to estimate default energy and emission rates and adjustments used by MOVES2004.

- **Validation:** The document "MOVES2004 Validation Results" presents the methodology used to validate MOVES2004 against top-down fuel sales data, and the results.
- **GREET Documentation:** The document "User Manual and Technical Issues of GREET for MOVES Integration", produced by Argonne National Laboratory, explains the integration of the GREET model into MOVES and illustrates its use.

In addition, MOVES2004 also contains online help documentation, accessible under the HELP menu in the MOVES interface.

What's Next

If MOVES is installed and operational on your computer, you may want to run the example→see **Executing Example MOVES2004 Run Specification**.

If you are uncertain about the installation→follow instructions in **Testing Your Installation**.

If you are not familiar with MOVES terminology and design→see the **Design Reference Manual** (listed in **Other Documentation and Online Help**).

If you have questions about the MOVES Graphical User Interface (GUI) or executing a run→see **Overview of MOVES RunSpec User Interface**.

If you are ready to begin using MOVES→see **Starting MOVES** .

Installation

Detailed instructions on installation are supplied by EPA with the MOVES2004 distribution package. This package consists of two programs (MySQL and MOVES2004), all other required software platform components, and both electronic and print Installation Guide documentation. Please refer to those materials first if MOVES2004 has not been installed on your computer.

System Requirements

The MOVES program is open source and written in JAVA and MySQL, but is designed to work only in a Microsoft Windows 2000 and later environment. Computer(s) used to run either of the MOVES application programs must have at least 128 MB of RAM, (256MB or more recommended). Execution run time performance is a constraint with MOVES so high speed processor(s), at least 1 GHz and preferably faster, are highly recommended.

The MOVESDefault database distributed with MOVES requires approximately 275 MB of disk storage. MOVES Worker and Output databases are also often voluminous, so several gigabytes of disk space should be available on all machines used to run either MOVES program. Serious users of MOVES2004 will want to use late-model, high-performance microcomputer systems.

MOVES may be operated on a single computer system or a network of computers. See the Installation Guide and/or Design

Reference Manual for more information about specific requirements and computer configuration.

Testing Your Installation

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

1. Check that the MySQL server is operating on all computer systems in the configuration that require it. This is done by opening the MS DOS prompt and changing the path to "C:\mysql\bin". After changing the path, the user should type 'mysql' at the prompt. The MySQL program should begin. If an error message appears, the MySQL program or server has not been successfully installed. On most versions of WINDOWS the MySQL server can be set up to run automatically as a system service. Detailed instructions as to how to do this are contained in the Installation Guide.
2. Start the MOVES2004 Worker program on one or more computers by double-clicking its program icon. On multiple-computer configurations these MOVES Worker programs are usually left running indefinitely. This program does not have to be on the same computer as the MOVES Master Program, but must have access to the shared file directory.

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

3. Start the MOVES Master/GUI program by double-clicking its icon. This master program may be installed on a single computer or on a computer network to allow several computers to run concurrently.
4. Execute the Example Run Specification (MOVES2004Example.mrs) as described in Section 3 of this manual.

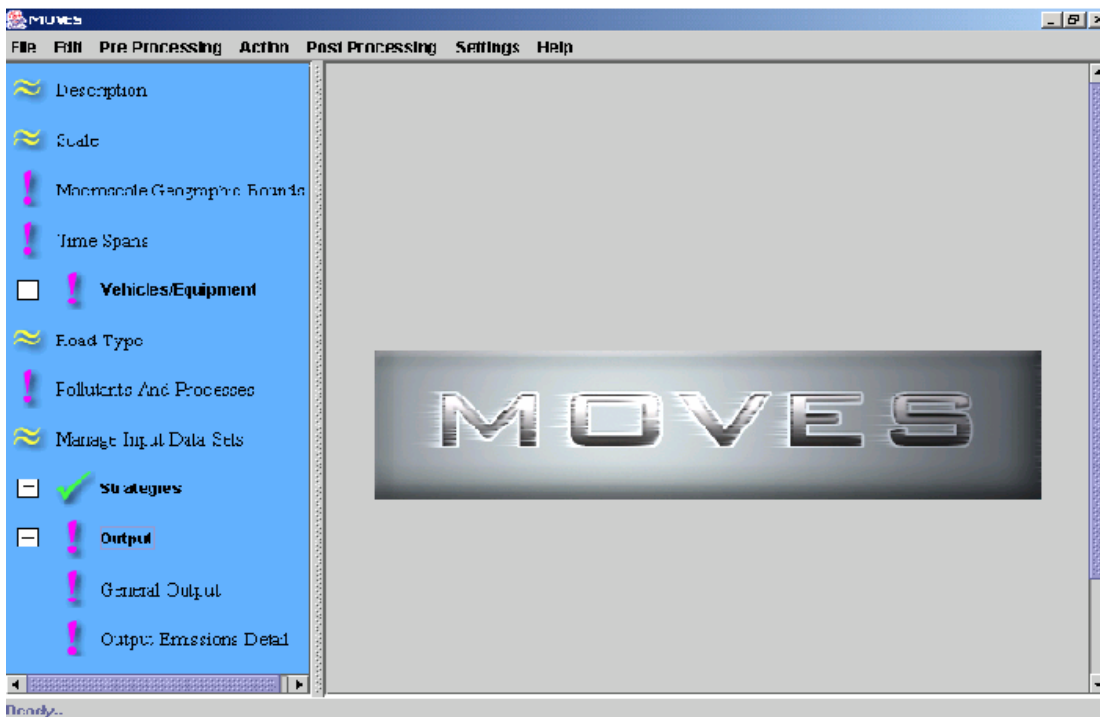
!TIP Be sure to create the required output database MOVES2004Example by going to the General Output screen and clicking on the button.

2. Using MOVES2004

MOVES2004 has a Graphical User Interface (GUI) to set up and operate a MOVES Run Specification (RunSpec). After an overview of the screen layout and navigation functions of the GUI, details on how to start MOVES and operate a RunSpec are provided.

Overview of MOVES RunSpec User Interface

The MOVES RunSpec User Interface (UI) consists of three parts: Main Menu Bar, Navigation List, and Detail Panel. The Menu Bar and Navigation List show on every screen with the Detail Panel changing as items on the Navigation List are selected.



Though not shown here, a progress indicator will also appear on the screen during execution of a run. The user can pause, resume, and cancel the run.

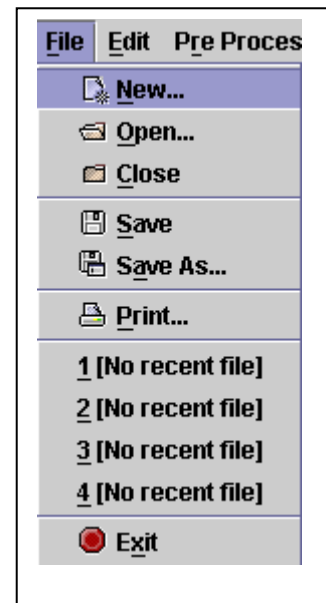
The MOVES model contains a “mouse hover” user help feature that is present throughout the model. To utilize it, the user simply places the mouse over a particular MOVES GUI icon and keeps it very still. A small help box should appear containing a brief text message. The text message will contain the readiness status of a navigation panel icon or a brief description of a detail panel icon.

Main Menu Bar

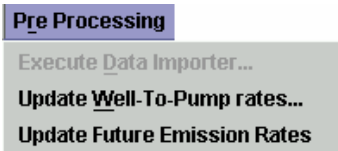
The Main Menu Bar runs across the top of the screen displaying seven drop down menus: File, Edit, Pre Processing, Action, Post Processing, Settings, and Help.

File provides a drop-down menu typical of those used when manipulating documents. Drag and click to select New, Open, Close, Save, Save As, Print, previously opened files, and Exit. Use these to create, load from disk, and save RunSpec objects. The Example RunSpec may be opened from this menu (see Section 3 for details).

EDIT provides the usual cut/copy/paste commands. These commands are only available in text entry sections of MOVES RunSpec.



PRE PROCESSING contains three items, of which only two are operable in the current version. Do not attempt to use EXECUTE DATA IMPORTER at this time. Select UPDATE WELL-TO-PUMP RATES... to execute the GREET model, or select Update Future Emission Rates to generate new future emission rates that can be added to a MOVES simulation using the Manage Input Data Panel (see Section “Specifying Additional Databases (Input Data Sets) (optional)”).



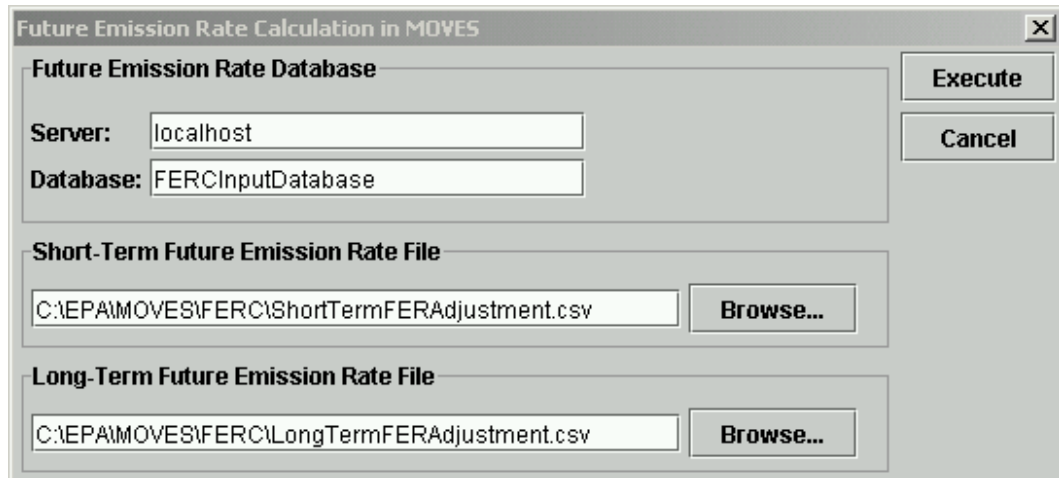
The UPDATE WELL-TO-PUMP RATES... will update the energy and emission rates for the Well-To-Pump process; i.e., energy consumed and emissions produced in the process of getting fuel from raw material to the pump. For example, this allows the user to account for emissions that are associated with the production and distribution of a gallon of fuel that is burned by an on-road vehicle. This feature is only required if the user wishes to change inputs on the well-to-pump energy and emission calculations from the defaults stored in the MOVES Default database. Selection of this option

will send the user into the GREET model GUI (See Appendix A, "Using GREET to update Well-To-Pump Energy and Emission Factors").

The UPDATE FUTURE EMISSION RATES menu item allows the user to create energy and emission rates for alternative fuel and advanced technology vehicles for model years 2001-2010, and for all vehicles for model years later than 2010. The menu item accesses a MOVES Pre-Processing program known as the "Future Emission Rate Creator" (FERC) that creates new emission rates from user supplied data. The resulting emission rates are created as a new MySQL database and data file which are input to the MOVES model through the Manage Input Data Panel (see Section "Specifying Additional Databases (Input Data Sets) (optional)" for complete details).

NOTE MOVES2004 does NOT provide default rates for alternative fuel and advanced technology vehicles for model years 2001-2010, or for any post-2010 model year vehicle. The user is therefore required to generate these rates using the FERC with inputs provided with the model, or customized by the user. Once the desired set of rates has been generated by the FERC, this step does not need to be repeated until an alternate set of future rates is desired.

If the UPDATE FUTURE EMISSION RATES menu item is chosen the MOVES window "Future Emission Rate Calculation in MOVES" will open. This panel allows the user to specify the server and MySQL database name in which the future emission rates that are generated will be placed. The user types these names in the text boxes on the FERC GUI panel. The panels "Short-Term Future Emission Rate File" and "Long-Term Future Emission Rate File" allow the user to browse and specify the name of the required future emission rate data files. The short term file pertains to rates for model years 2001 – 2010; the long term file pertains to rates for later model years.



NOTE The user cannot name the database containing the future emission rates "MOVESDefault" or

“MOVESExecution”. It is also highly recommended that the server name remain “localhost”.

MOVES2004 supplies examples of the “Short-Term Future Emission Rate File” and “Long-Term Future Emission Rate File”, found in the “FERC” directory by using the Browse buttons in the GUI (this directory is in the MOVES directory folder path specified at the time of the MOVES installation. This is most likely at “C:\MOVES ...”). These files are based on analysis discussed in detail in the report “MOVES2004 Energy and Emissions Inputs”, and can be used directly in the FERC. Alternately, users wishing to customize the contents of these tables can use these as templates, following the instructions in the EPA document “Draft MOVES2004 Software Design Reference Manual”.

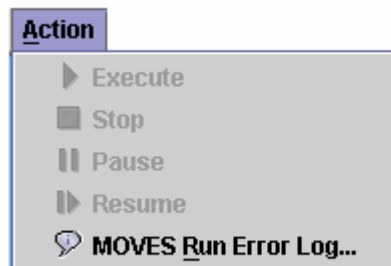
After the appropriate inputs are made the FERC GUI panel, the “Future Emission Rate Calculation in MOVES” calculation is begun by pressing the “Execute” button on the FERC GUI screen. If the Pre Processing is successful, energy and emission rates for alternative fuel and advanced technology vehicles for model years 2001-2010 and for all vehicles for model years later than 2010 will be placed in the EmissionRate table in the MySQL database specified by the user. Depression

of the "Cancel" button on the FERC GUI clears all of the FERC GUI inputs and closes the FERC panel.

NOTE The database created by the FERC MUST be specified on the "Manage Input Data Sets" panel of the RunSpec in order to be included in the MOVES run. Users who wish to perform multiple MOVES runs using alternate sets of future energy and emission rates should consider specifying a different database for each set of rates, to be used for subsequent MOVES runs as desired.

NOTE The Pre Processor Future Emission Rates Calculation in MOVES does not require any other inputs to MOVES panels or a valid MOVES Runspec. Essentially, it can be viewed by the user as a stand-alone program that is built inside of the MOVES structure.

Action provides a drop-down menu with the choices Execute, Stop, Pause, Resume, and MOVES Run Error Log.... The first four are actions, the last item requests a display to pop up. MOVES can only be executed if all of the required RunSpec inputs have been satisfied. Until they are satisfied, the EXECUTE item will remain grayed out.



After the user satisfies the RunSpec input requirements (all check marks in the navigation panel must be set to green checks or yellow tildes – see “Navigation List” for an explanation of the green checks and yellow tildes), click EXECUTE to execute the MOVES simulation. Click STOP or PAUSE items in the ACTION menu to stop or pause the execution of the MOVES program. These two items will only be active if the MOVES program is running. The user may also resume a paused MOVES simulation by clicking the RESUME item.

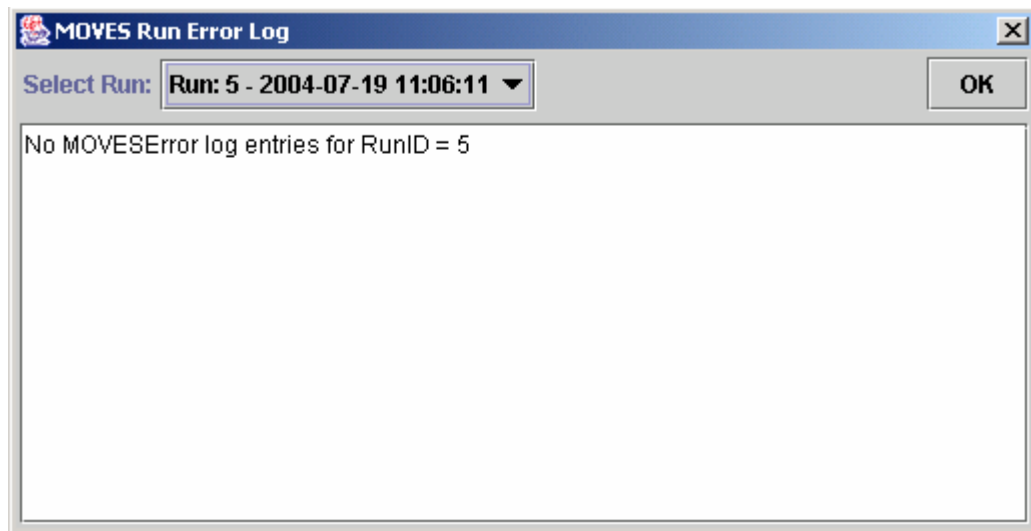
NOTE A program that has been stopped cannot be resumed.

Click MOVES RUN ERROR LOG... to display the MOVES Run Error Log panel.

NOTE This panel will only appear if a MOVES Output database has been created.

Typically, the user should select the MOVES RUN ERROR LOG ... immediately after the MOVES program has completed its execution. The panel (shown following) will contain either a message stating that the run was

successful and no errors occurred, or it will report a brief error diagnostic. The example shows a successful run, indicating the run number (Run 5) and the time and date of the run. The user can also view the Run Error Log for the error status of previous runs by clicking the “down arrow” on the right side of the Select Run button.

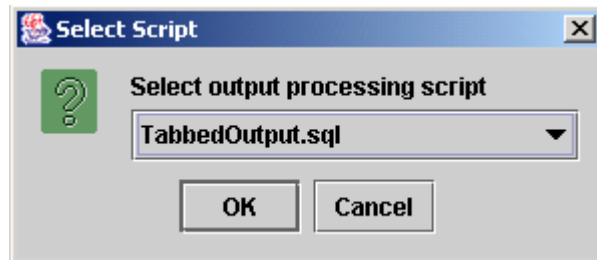


POST PROCESSING has one item, Run MySQL Script on Output Database. This feature is used after a successful execution of the MOVES simulator to further process the MOVES simulator results into more aggregate or easy to use forms. The MOVES Post Processing item consists of a set of MySQL Scripts that process the MOVES output databases (stored in the MySQL database format).

Post Processing

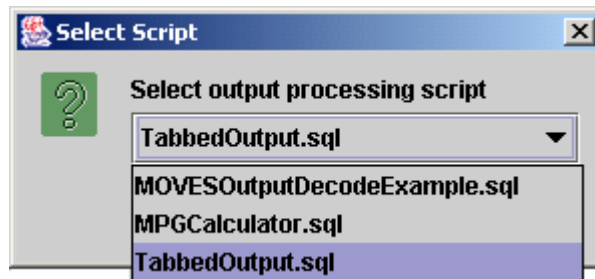
Run MySQL Script on Output Database

Click "Run MySQL Script on Output Database" to open the SELECT SCRIPT box, as shown here.



Select the "OK" button to execute the script shown in the center scroll down list. In this example, it is the "TabbedOutput.sql" script.

Click on the arrow in the scroll down box to view all the available scripts, as shown here.



Select a post-processing script by clicking to highlight it.

Currently three scripts are available for use. 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 second script "MPGCalculator.sql" calculates the miles per gallon (MPG) fuel economy and the gasoline equivalent MPG fuel economy from the total energy results in the MOVES Output database. The calculation requires that "fuel type" be distinguished in the MOVES output. Otherwise, the results will be meaningless.

The third script "TabbedOutput.sql" takes the three MOVES MySQL output tables (MOVESOutput, MOVESActivityOutput and MOVESRun) and turns them into tabbed 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.

!TIP The Post Processing feature of the MOVES simulator is also designed to provide a mechanism to allow the user to write MySQL scripts that transform standard MOVES output into results that more closely fulfill specific modeling needs. Specific instructions on writing a MySQL script are beyond the scope of this document. However, the script must be a text file containing the appropriate MySQL SQL commands (SQL means Standard Query Language). The name of the file must

have the suffix “.sql”. When complete the “sql” text file must be placed in the MOVES program folder

```
“MOVES”  
  subfolder “Database”  
    subfolder “OutputProcessScripts”
```

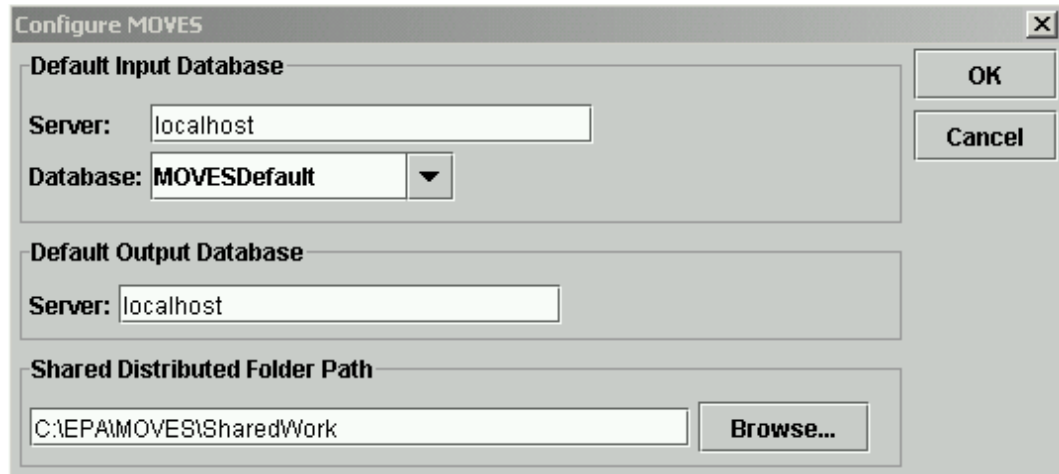
i.e., Path
C:\...\MOVES\Database\OutputProcessScripts

The name of the added script will appear in the scroll down menu box of the “select scripts” window shown above.

If the user is interested in creating their own post processing scripts and is familiar with the Standard Query Language (SQL), they should carefully study the MOVES Output database and the example scripts.

SETTINGS drop-down menu has one item. Select CONFIGURE MOVES if you want to:

- Select the MOVES database. In most cases choose “MOVESDefault.”
- Select the Server in which to access the database. In most cases, choose “localhost.”
- 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. Browse your 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.

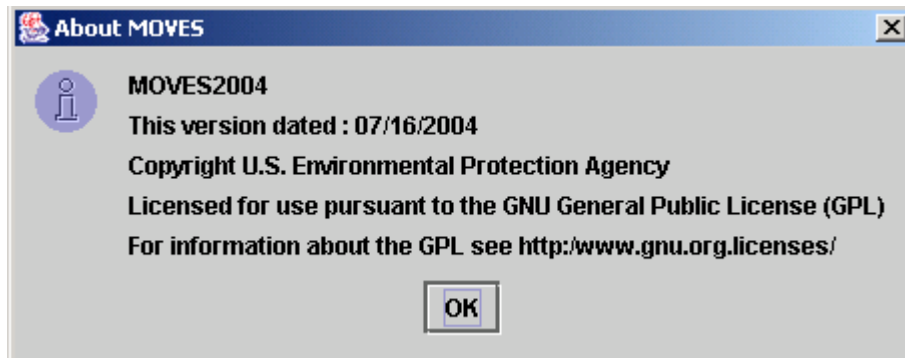


HELP provides access to the online MOVES User's Guide and general information about MOVES. The online help contains the material in this User Guide and the MOVES Software Design Reference Manual.



Click on "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 their personal use. For specific details regarding the GNU license, the user should consult the website <http://www.gnu.org/licenses/>



Navigation List





The **Navigation List** appears in blue on the left half of the screen (shown in first screen example in "Using MOVES.")

Selecting an item from the navigation list will place that item's detailed user interface into the detail panel on the right of the screen.

The RunSpec navigation list depicts a tree-like structure of areas of the RunSpec's information. Some sections, such as "Vehicles/Equipment" and "Output," contain subsections. These sections are shown with an icon (or) that allows a list of subsections to be expanded or collapsed.

NOTE Sections may be visited in any order and selections on them have no lasting effects until the run specification (RunSpec) is saved or the model is executed.

RunSpec navigation list items are shown with an icon that indicates the completeness of the RunSpec in that section, as shown in the table following:

Icon	Meaning
	Needs additional user supplied data.
	Sufficiently filled in to run.
	Default data present, but otherwise sufficiently filled in to run.
	<u>TREE CLOSE/EXPAND</u>

!TIP The icons shown on the sample UI in this document are not necessarily indicative of which sections/subsections will have default data available.

!TIP RunSpec cannot be executed until all necessary data is supplied.

Detail Panel

Detailed UI for each input and output item are provided in the **detail panel** that occupies most of the screen. These are explained in the appropriate section describing the function and operation of the UI.

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/disabled based upon selection in their associated list box. For example, a "Delete" button will be disabled until a selection is made in its list box.

Starting MOVES

After installation, MOVES may be started with the following steps.

1. Double-click the MOVES2004 Worker program icon.

🔗 NOTE On multiple-computer configurations these MOVES Worker programs are usually left running indefinitely. This program does not have to be on the same computer as the MOVES Master Program, but must have access to the shared file directory.

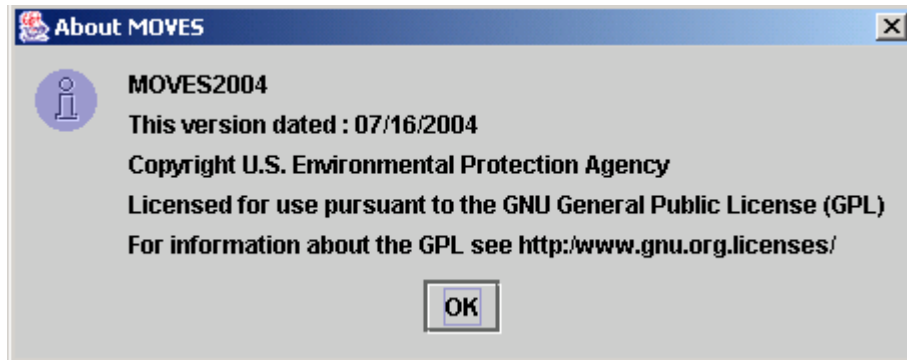
⚠ CAUTION! It would be detrimental to performance to operate more than one copy of the worker program on a single computer.

2. Double-click the MOVES Master/GUI program icon.

🔗 NOTE This master program may be installed on a single computer or on a computer network to allow several computers to run concurrently.

3. An ABOUT MOVES panel will appear in the center of the GUI screen. Click the "OK" button to continue with the MOVES simulation.

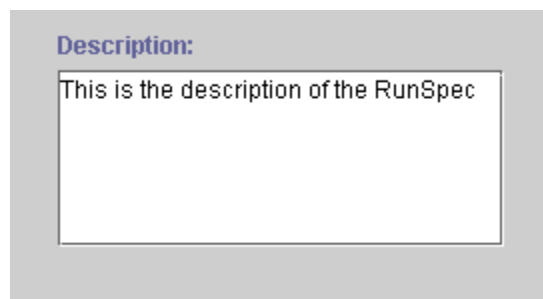
🔗 NOTE This panel identifies the particular version of MOVES that is in use via the version date, states the EPA copyright, and provides a link for the GNU General Public License (GPL) website (for further information on GPL see the "About MOVES" section of this document). This panel will appear only once upon start-up.



4. The MOVES RunSpec User Interface (UI) will open and input information may be entered. Entries may be made in any order, but a RunSpec cannot be executed until all the required information is provided.

Describing RunSpec

Select **DESCRIPTION** , the first item on the Navigation List, to open a scrollable text window that allows the user to give the RunSpec a particular textual description 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 and will not be reported anywhere else.



Up to 5000 characters of text may be entered to describe the RunSpec. In the above example "This is the description of the RunSpec" is the text. Text entered has no effect on the results from the MOVES program.

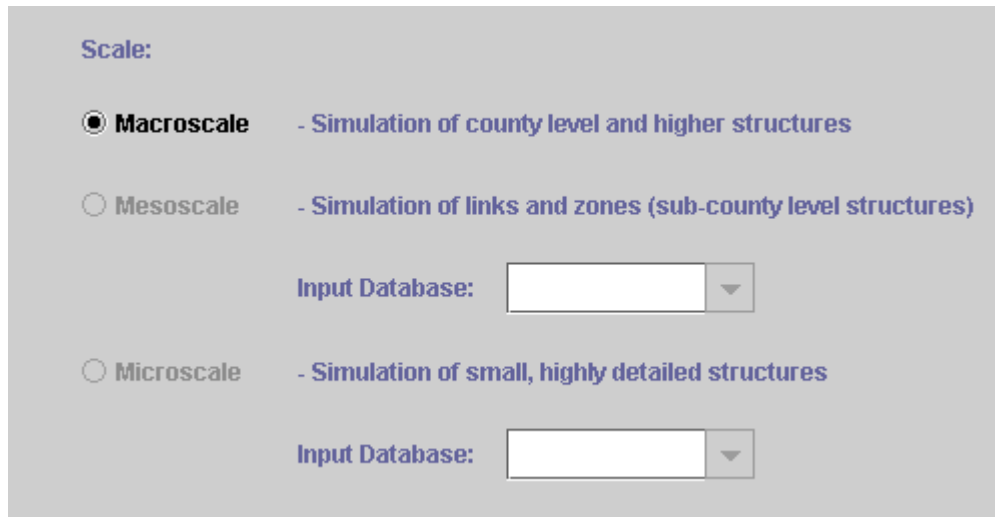
!TIP The default Description is blank and the MOVES model can be run without a Description.

!TIP Do not use non-text characters (i.e., "&") in your Description, since these characters are sometimes misinterpreted and can interfere with the proper operation of MOVES.

Selecting Scale

Select **SCALE** on the Navigation List to choose the scale level at which the model will operate.

NOTE No more than one level may be chosen.



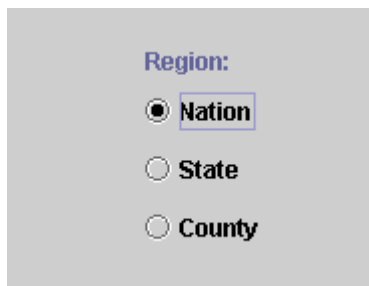
The screenshot shows a user interface for selecting a scale. It is titled "Scale:" and contains three radio button options. The first option, "Macroscale", is selected and has a description: "- Simulation of county level and higher structures". Below it is an "Input Database:" label followed by a dropdown menu. The second option, "Mesoscale", is unselected and has a description: "- Simulation of links and zones (sub-county level structures)". The third option, "Microscale", is unselected and has a description: "- Simulation of small, highly detailed structures". Below it is another "Input Database:" label followed by a dropdown menu.

The chosen scale determines which geographic panel (Macroscale Geographic Bounds) will be presented to the user and may affect the availability of other panels as well. Currently, only the Macroscale option is available and, so, is the default selection.

The other two Scales, "Mesoscale" and "Microscale," currently cannot be enabled by the user. They will be enabled in future versions of MOVES.

Selecting Macroscale Geographic Bounds

Select **MACROSCALE GEOGRAPHIC BOUNDS** to open the initial Macroscale Geographic Bounds Panel, as shown below.



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

CAUTION! Geographical Data Pre-Aggregation 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) all underlying data that is a function of geography (temperatures, road types, etc) into more singular values and execute the "National" or "State" run as a single county run. 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.

🔗 NOTE An alternative method of computing a state simulation is to select "County," then a particular state, and finally all of the individual counties in the particular 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 the alternate method is used to run state and nation at the county level, execution times will be potentially very long and large runs would likely require multiple-computer processing.

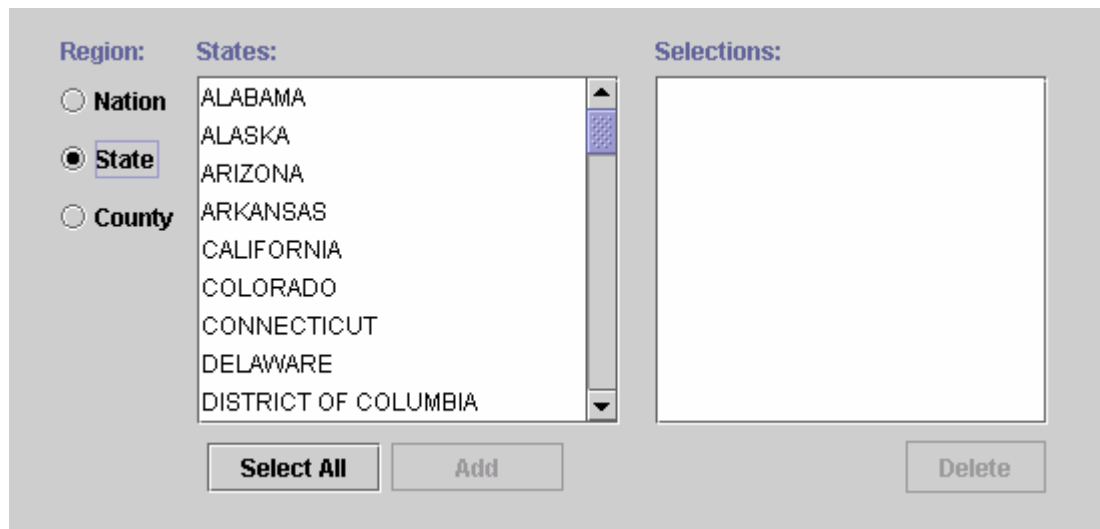
The following information goes through the three options and input needed for each button.

Nation

Choose "*Nation*" to run scenarios and compute emission inventory results for all counties/parishes in the United States and Territories. If the "*Nation*" button is selected, this completes the Macroscale Geographic Bounds input requirement and no further Macroscale Geographic Bounds windows will appear. The "*Nation*" button is generally chosen if the user wishes to create a national U.S. emission inventory.

State

Choose “*State*” to run scenarios and compute emission inventories for all specific areas of the United States and Territories. If the “*State*” button is selected a new bounds panel will show in the Detail Panel, as shown below.



The screenshot shows a software interface with three main sections: "Region:", "States:", and "Selections:". Under "Region:", there are three radio buttons: "Nation", "State" (which is selected), and "County". The "States:" section is a list box containing the following states and territories: ALABAMA, ALASKA, ARIZONA, ARKANSAS, CALIFORNIA, COLORADO, CONNECTICUT, DELAWARE, and DISTRICT OF COLUMBIA. Below the list box are two buttons: "Select All" and "Add". The "Selections:" section is an empty rectangular box. Below the "Selections:" box is a "Delete" button.

This panel shows three parts (Region, States, and Selections) and buttons for choosing the appropriate selections. The “*State*” button will already be selected. To choose the desired state(s):

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.

Click the “*Select All*” button to choose all US states and territories available.

Highlight a previously selected state in SELECTIONS box.

Click the “*Delete*” button to remove the highlighted state.

🔗 **NOTE** The “Add” and “Delete” buttons are inactive if no states are highlighted.

🔗 **NOTE** If the *State* button is selected in the Region box, the model cannot be run without a State selection. A small panel at the bottom will remind the user to select a state. This panel will disappear after the user makes the correct selection.

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.

Region:	States:	Counties:	Selections:
<input type="radio"/> Nation	KENTUCKY	MICHIGAN - Schoolcraft County	MICHIGAN - Washtenaw County
<input type="radio"/> State	LOUISIANA	MICHIGAN - Shiawassee County	
<input type="radio"/> County	MAINE	MICHIGAN - St. Clair County	
	MARYLAND	MICHIGAN - St. Joseph County	
	MASSACHUSETTS	MICHIGAN - Tuscola County	
	MICHIGAN	MICHIGAN - Van Buren County	
	MINNESOTA	MICHIGAN - Washtenaw County	
	MISSISSIPPI	MICHIGAN - Wayne County	
	MISSOURI	MICHIGAN - Wexford County	

This panel shows four parts (Region, States, Counties, and Selections) and buttons for choosing the appropriate selections. The “County” button will already be selected. Choose the desired county(ies) as follows:

Highlight the desired state. Individual counties will then appear in the COUNTIES box. In this example, the State of Michigan is highlighted and the Michigan counties appear. Highlight the desired county by scrolling to and clicking on the name of a particular county. Then click the “Add” button to move the desired state-county combination to the SELECTIONS box.

Click on the “Select All” button to choose all counties in the highlighted state.

Highlight a previously selected state-county combination and click on the “Delete” button to remove it from the SELECTIONS box.

NOTE The “Add” and “Delete” buttons are inactive if no counties are highlighted.

NOTE If the *County* button is selected in the Region box, the model cannot be run without a State-County selection. A small panel at the bottom will remind the user to select a state and county combination. This panel will disappear after the user makes the correct selection.

Macroscale Geographic Bounds Requirements

Please select a state and county.

Selecting Time Spans

MOVES 2004 can be set to model for specific time periods by selecting **Time Spans** on the Navigation List. This will open the Time Spans panel as follows.

The screenshot displays the 'Time Spans' panel with the following sections:

- Time Aggregation Level:** Radio buttons for Year, Month, Day, and Hour. The 'Hour' option is selected.
- Years:** A 'Year:' dropdown menu set to '2000' with an 'Add' button. Below it, a 'Year:' text input field contains '2000' and a 'Remove' button.
- Months:** Checkboxes for each month from January to December. 'January' is checked. 'Select All' and 'Clear All' buttons are at the bottom.
- Days:** Checkboxes for each day of the week (Monday through Sunday). All days are checked. 'Select All' and 'Clear All' buttons are at the bottom.
- Hours:** 'Start Hour:' and 'End Hour:' dropdown menus. 'Start Hour' is set to '10:00 - 10:59' and 'End Hour' is set to '11:00 - 11:59'. 'Select All' and 'Clear All' buttons are at the bottom.

This panel is divided into five sections with boxes, buttons, or drop-down menus in each to select specific aggregate levels, years, months, days, and hours.

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.

🔗 NOTE 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) all underlying MOVES internal data that are segregated by hour (i.e., temperatures, VMT distributions, etc) into more singular sets of values 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 will be a set because the temperatures will not necessarily be averaged by geography), and perform the simulation with these average values.

🔗 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" inputs. After the initial automatic selection, the user may de-select particular hours, 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 all of the "Days" except "Monday" the final emission result will represent a monthly aggregation that is all Mondays in the chosen month (approximately four days).

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

Years

Select a calendar year(s) in the **YEARS** section and click on the "Add" button. Select a year by holding the black triangle to see a drop-down list of calendar years, ranging from the 1999 through the 2050 calendar year, and highlight 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. 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.

🔗 NOTE MOVES2004 does NOT provide default energy and emission rates required to model calendar years 2001 and later; in order to produce meaningful results for these years, the user is required to generate these rates using

the Future Emission Rate Creator (FERC), as discussed in Section 2.

Months

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

Days

Click one or more individual days of the week to model to select the appropriate boxes in the DAYS section. At least one 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 days of the week.

Hours

The default time resolution for MOVES2004 is hourly, with hours expressed in military time ranging from midnight – 12:59 am (expressed as 00:00 – 00.59) to 11 pm - 11:59 pm (23:00 – 23:59). Click one or more individual hours of the day to model to select the appropriate Start and End hours from the list. The model cannot be run without a selection on this panel and no default value appears. Click "*Select All*" button to select all of the hours of the day. Similarly, the "*Clear All*" button removes all of the previously 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** It is not possible for a start and end time to include more than one day, for example, from 11:00 PM on Friday to 01:59 AM on Saturday. It is also not possible to run the model for months that overlap years. 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.

Selecting Vehicles/ Equipment

To select which on road vehicles are to be modeled, click on **VEHICLES/EQUIPMENT** in the Navigation List. This will open the ON ROAD VEHICLE EQUIPMENT detail panel. Two scroll down lists (*Fuels* and *Source Use Types*) appear to the left of this panel to allow distinct dimensions for the vehicles/equipment.

The screenshot displays a software interface for selecting vehicle equipment. It features three main columns: 'Fuels', 'Source Use Types', and 'Selections'. The 'Fuels' column lists various fuel types, with 'Gasoline' highlighted. The 'Source Use Types' column lists vehicle categories, with 'Passenger Car' highlighted. The 'Selections' column shows the resulting combination 'Gasoline - Passenger Car'. Below the lists are three buttons: 'Select All' (under Fuels), 'Select All' (under Source Use Types), and 'Delete' (under Selections). A fourth button, 'Add Fuel/Type Combinations', is positioned at the bottom center.

Fuels:	Source Use Types:	Selections:
Compressed Natural G...	Combination Commerci...	Gasoline - Passenger Car
Diesel Fuel	Combination Delivery Tr...	
Electricity	Interstate Bus	
Ethanol (E85 or E95)	Light Commercial Truck	
Gaseous Hydrogen	Motorcycle	
Gasoline	Motorhome	
Liquid Hydrogen	Passenger Car	
Liquid Propane Gas (LP...	Passenger Truck	
Methanol (M85 or M95)	Refuse Truck	

Select All **Select All** **Delete**

Add Fuel/Type Combinations

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

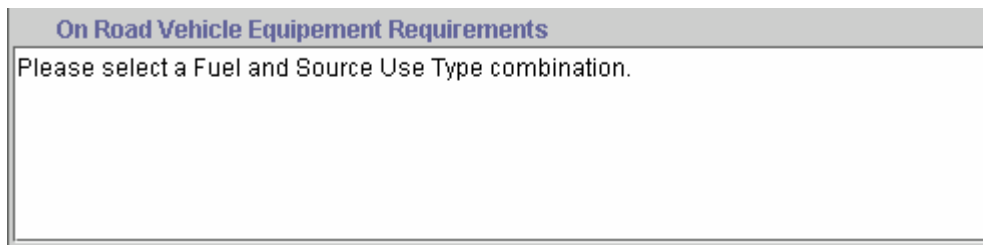
Click and highlight the SOURCE USE TYPES choice or click the *"Select All"* button to choose all the choices.

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

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

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

🔗 **NOTE** On Road Vehicle Equipment Requirements Panel contains a small panel 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.



Selecting Road Type

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

The screenshot shows a software interface for selecting road types. On the left, under the heading "HPMS Road Types:", there is a list of road types: Off-Network, Rural Interstate (highlighted), Rural Local, Rural Major Collector, Rural Minor Arterial, Rural Minor Collector, Rural Principal Arterial, Urban Collector, and Urban Freeway/Expressway. Below this list are three buttons: "Select All", "Add", and "Delete". On the right, under the heading "Selected Road Types:", there is a larger box containing the text "Rural Interstate".

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

1. Click and highlight the desired HPMS ROAD TYPES from the scroll down list.
2. Click "Add" button to moves choice to the SELECTED ROAD TYPES box.
3. Click "Delete" button to clear the highlighted selections.
4. Click "Select All" button to select all HPMS ROAD TYPES in the scroll down list.

🔗 NOTE The scroll down list of HPMS ROAD TYPES is the complete list of roadway types present in the underlying MOVES database.

🔗 NOTE “Off Network” is automatically selected when start or extended idle processes are selected on the “selecting pollutants and processes” panel.

⚠️ CAUTION! Entries to the HPMS ROAD TYPES list are not restricted to only those roadway types present in the particular geographical range selected by the user in the MACROSCALE GEOGRAPHIC BOUNDS Panel. Hence, some selected road types may show zero energy consumption, emissions and distance if the road types don’t exist in the selected geographic area.

Selecting Pollutants and Processes

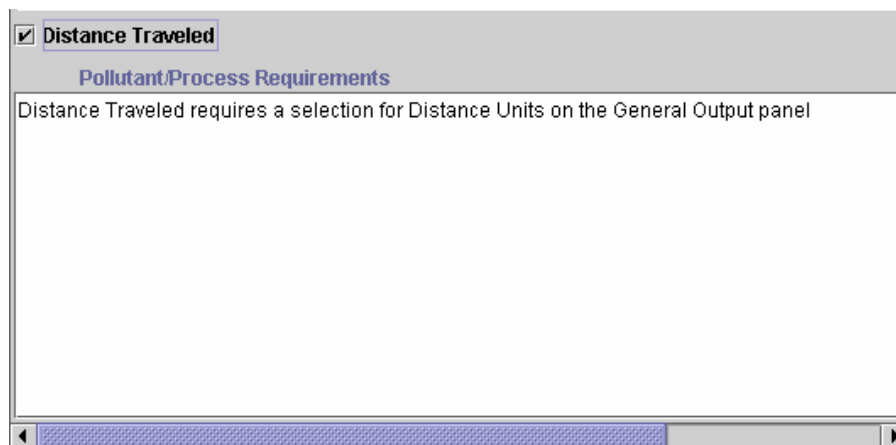
Click **Pollutants and Processes** on the Navigation List to select which pollutants and processes to use in the run.

	Extended Idle Exhaust	Running Exhaust	Start Exhaust	Wall-To-Pump
Fossil Fuel Energy Consumption	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Methane (CH4)		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Nitrous Oxide (N2O)		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Petroleum Energy Consumption	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Total Energy Consumption	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

1. Click on desired box. A will appear indicating that combination will be calculated.
2. Check as many boxes as desired. However, at least one must be checked to produce a valid MOVES RunSpec input.

NOTE The MOVES model allows the user to select most combinations of four processes and five pollutants. In some cases, pollutant – process combinations must be included in order for others to be calculated. MOVES will alert the user if an incomplete group is selected.

3. If desired, select the DISTANCE TRAVELED check-box to have the MOVES model compute and report the distance traveled by



vehicles in the selected geographic, roadway, or time strata.

NOTE The DISTANCE TRAVELED box will be 'grayed out' if no Running Exhaust Process is chosen in POLLUTANTS AND PROCESS selection panel. It also requires a selection for DISTANCE UNITS on the GENERAL OUTPUT panel.

NOTE The POLLUTANT/PROCESS REQUIREMENTS subpanel alerts THE USER if they make an invalid or incomplete pollutant / process choice. Certain pollutant / process combinations require other pollutant / process boxes to be checked before a valid RunSpec can be created.

The four processes and five pollutants are described briefly:

Processes

- a. Extended Idle Energy Consumption / Exhaust — Energy consumed or exhaust emissions produced from a vehicle while idling for an extended period, typically a heavy-duty (i.e. combination long-haul) truck idling overnight in a parking lot. Idle operation while the vehicle is in traffic (i.e., waiting at a traffic light) is not considered Extended Idle.
- b. Running Energy Consumption / Exhaust — Energy consumed or exhaust emissions produced from a vehicle while it is operating on-road.
- c. Start Energy Consumption / Exhaust — Energy consumed or exhaust emissions produced from a vehicle during its cold

and hot start operation. Defined as the *incremental* energy or emissions that result from start operation.

- d. Well-to-Pump—The energy consumed or exhaust emissions produced in the extraction, processing and distribution of a vehicle's fuel.

Pollutants

- a. Fossil Fuel Energy Consumption—This pollutant is in units of energy (i.e., joules or BTU). For the pump-to-wheel process, it is the vehicle energy consumption that is contributed by burning fossil fuel sources. Biodiesel and other such sources are excluded. For the well-to-pump process, it is the amount of fossil-based energy used in the extraction, processing and distribution of a vehicle's fuel.
- b. Methane—This pollutant is the total amount of methane (CH₄) emitted from vehicle sources in mass units.
- c. Nitrous Oxide—This pollutant is the total amount of nitrous oxide (N₂O) emitted from vehicle sources in mass units.
- d. Petroleum Energy Consumption—This pollutant is in units of energy (i.e., joules or BTU). It is the vehicle energy consumption that is generated by burning petroleum fuel sources. Non-petroleum fuel sources such as natural gas (Fischer-Tropsch diesel) and other such sources are excluded. For the well-to-pump process, it is the amount of petroleum-based energy used in the extraction, processing and distribution of a vehicle's fuel.
- e. Total Energy Consumption—This pollutant is in units of energy (i.e., joules or BTU). It is the vehicle energy consumption

that is generated by all fuel sources. For the well-to-pump process, it is the total energy used in the extraction, processing and distribution of a vehicle's fuel.

A DISTANCE TRAVELED checkbox and the POLLUTANT/PROCESS REQUIREMENTS panel appear at the bottom of the POLLUTANT AND PROCESSES panel.

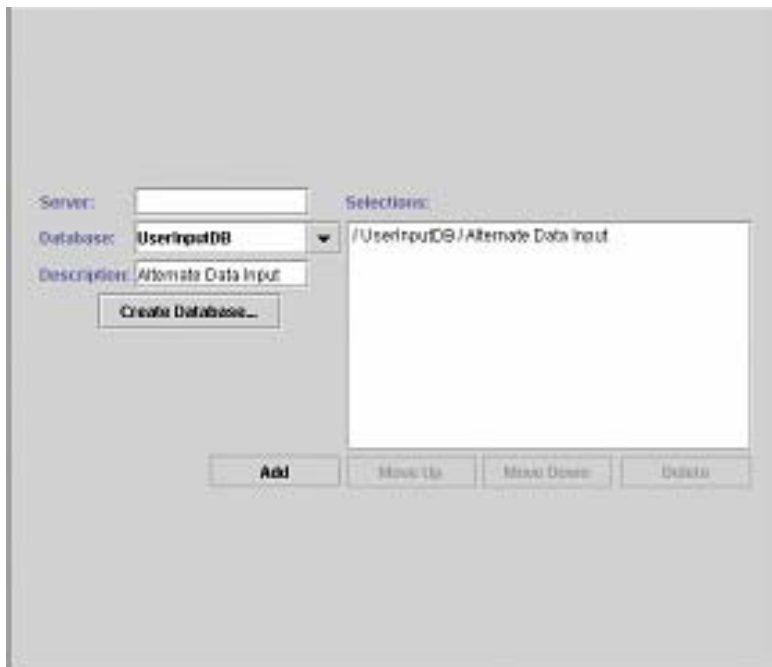
Check the DISTANCE TRAVELED check box to have the MOVES model compute and report the distance traveled by vehicles in the selected geographic, roadway, or time strata. This box will be "grayed out" if no Running Exhaust Process is chosen.

!TIP A selection for DISTANCE UNITS on the GENERAL OUTPUT panel is also required for this computation.

A message appears in the POLLUTANT/PROCESS REQUIREMENTS panel to alert the user of an invalid or incomplete pollutant/process choice. Certain pollutant/process combinations require other pollutant/process boxes to be checked before a valid RunSpec can be created.

Specifying Additional Databases (Input Data Sets) (optional)

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



The screenshot shows a software interface for managing input data sets. It features a 'Server' text field, a 'Database' dropdown menu currently set to 'UserInputDB', and a 'Description' text field containing 'Alternate Data Input'. Below the description is a 'Create Database...' button. To the right of these fields is a 'Selections' list box containing the entry '/UserInputDB/Alternate Data Input'. At the bottom of the interface are four buttons: 'Add', 'Move Up', 'Move Down', and 'Delete'.

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 MOVESDEFAULT database and the individual fields in the tables must have identical lengths and types as MOVEDEFAULT. One way for the user to achieve this desired schema is to start with an empty copy of the MOVESDEFAULT table(s) that is to be the alternate table and insert / modify / replace it with the new data. For details regarding the MOVES database schema see the MOVES2004 Software Design Reference Manual.

1. Click the "Create Database" button to select a new database on an existing local or remote server.

NOTE A database added through this feature can consist of one or more MOVES database tables. The example in the figure above shows the addition of an alternate database called "UserInputDB" (with a descriptive name of "Alternate Data Input") being added as a MOVES input. The unspecified Server name implies that the database resides in the user's local computer in the MySQL Data folder.

2. Click "Add" button to add the 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.

TIP The data in these databases will "overlay," that is augment or fully / partially replace, data from the

MOVES default database for the duration of the model run.

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

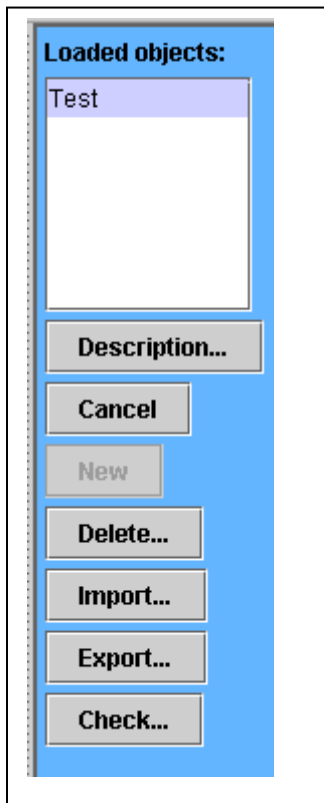
!Tip The order in which these databases are applied to the default data is important. If two or more alternate databases are specified in the Selection list, the one that appears last in the list will replace any subsequent ones in cases where the internal tables and records of each database are the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

4. Click the “*Delete*” button to delete selections.

The Mesoscale and Microscale options and the input databases specified on the MOVE model SCALE panel are not yet implemented. However, when they become available, the input databases on the SCALE panel will always be read first followed by the selections on the MANAGE INPUT DATA SETS panel.

Selecting Strategies

Select **STRATEGIES** on the Navigation List to open the STRATEGIES MANAGEMENT PANEL where strategies can be added, imported, exported, or deleted. This panel consists of two subpanels. All have the same left side panel (shown and explained following) with a detail panel specific to the strategy on the right side.



Click buttons on the right side of the panel to manipulate the strategies needed for each RunSpec, as detailed in the following. The strategy name will appear in the window under LOADED OBJECTS.

Adding Objects

Click on the "New" or the "Import" button to add a strategy to your RunSpec.

When clicking the "New" button, a new object will be created, added to the RunSpec, and selected in the list of objects.

When clicking the "Import..." button, an XML must also be selected from a previous "Export" of a Strategy.

NOTE These files are **not** the same files as those saved as whole RunSpecs, but instead hold only the settings pertinent to exactly one instance of a strategy.

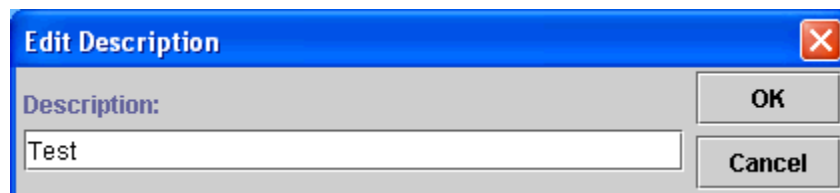
CAUTION! Some types of STRATEGIES can only have one instance within any RunSpec. A prime example of this is the [Alternative Vehicle Fuels & Technologies \(AVFT\) Strategy](#). Such strategies do not allow "New" to be used without first clicking "Delete" and will cause "Import" to overwrite any existing object.

Deleting Objects

Highlight loaded object to be deleted from the RunSpec. Click "Delete..." button. MOVES will prompt for confirmation before actually removing the object.

Editing Objects

The description of a Strategy object is displayed in the list of loaded objects and is set by selecting the object in the list then clicking the "Description..." button. A dialog requesting the description is displayed:



Descriptions can never be completely blank (they wouldn't show on screen then), so entering a blank description results in the phrase (*default*) being displayed instead. Do not confuse this default description to imply that the strategy contains only its default data.

Each strategy provides a GUI for editing its details. That GUI is displayed to the right of the STRATEGIES MANAGEMENT PANEL and is changed as objects are selected in the list of loaded strategy objects. Once an object is selected, details for it can be immediately edited.

Given the potentially complex data entry for a strategy, strategy GUIs provide a mechanism to undo a set of edits. Pressing the *Cancel* button will undo any edits made within a strategy's GUI since the strategy was last created, loaded, or imported. Be careful, as this can result in the loss of a great deal of data especially if you are unsure about what state your data was in at your last load. If you have a long data entry session to perform, you may wish to consider periodic *Export*'s of the data to provide finer-grain control of your rollback points.

The "*Check*" button can be used to check inputs to a Strategy. Clicking this button will display a popup message giving details about the reasons behind a "Not Ready" exclamation point.

Saving and Transferring Objects

As is implied by the above paragraphs, STRATEGIES can exist both inside and outside of a RunSpec. i.e. they are independent of the RunSpec file and can be re-used from one RunSpec to the next. This feature is helpful when a user wishes to create a large set of RunSpecs but use a consistent set of Strategy settings for each.

The “*Import*” and “*Export*” buttons allow transfer of XML-based files that contain exactly one Strategy outside of a RunSpec. Once such a file is imported, any changes made within the GUI are stored within the RunSpec and have no affect upon the originally imported file (unless an export is made to that file of course).

Using the FILE menu option to save and open RunSpecs automatically saves and loads the Strategies embedded within the RunSpecs and does not require in any way the files used to import those strategies.

Varying Vehicle Fuels and Technologies

To vary the expected fractions of vehicle fuels and technologies by model year, click the ALTERNATIVE VEHICLE FUELS AND TECHNOLOGIES (AVFT) STRATEGY Panel under Strategies on the Navigation List. This will open the AVFT screen.

The screenshot shows the AVFT screen interface. On the left is a 'Loaded objects:' panel with a 'Test' entry and buttons for 'Description...', 'Cancel', 'New', 'Delete...', 'Import...', 'Export...', and 'Check...'. The main area is titled 'Source Type:' and has a dropdown menu set to '21 Passenger Car', along with 'Normalize' and 'Add Model Year' buttons. Below this is a table with the following data:

	Gasoline Conventional Internal Combustion	Diesel Fuel Conventional Internal Combustion	Advanced Gasoline >>	Advanced Diesel >>	Alternative Fuel >>	Sum
2000	99.70%	0.08%	0.18%	0.00%	0.04%	100.00%
2001	99.70%	0.08%	0.18%	0.00%	0.04%	100.00%
2002	99.51%	0.07%	0.38%	0.00%	0.04%	100.00%
2003	99.10%	0.10%	0.77%	0.00%	0.04%	100.00%
2004	96.37%	0.18%	3.41%	0.00%	0.04%	100.00%
2005	95.95%	0.19%	3.83%	0.00%	0.04%	100.00%
2006	95.17%	0.19%	4.60%	0.00%	0.04%	100.00%
2007	94.11%	0.23%	5.62%	0.00%	0.03%	100.00%
2008	92.95%	0.24%	6.78%	0.00%	0.03%	100.00%
2009	90.86%	0.24%	8.85%	0.00%	0.03%	100.00%
2010	88.94%	0.29%	10.38%	0.36%	0.04%	100.00%
2011	86.85%	0.28%	12.47%	0.37%	0.04%	100.00%
2012	84.84%	0.28%	14.18%	0.66%	0.04%	100.00%
2013	82.96%	0.29%	16.02%	0.69%	0.05%	100.00%
2014	81.09%	0.29%	17.72%	0.86%	0.05%	100.00%

The left side of the screen is actually common to all Strategies and is referred to as the STRATEGIES MANAGEMENT PANEL.

Fundamentally, the AVFT strategy allows alternative entries for the FuelEngFraction MOVES database table. This table stores the fractions of engine and fuel technologies present within each model year for each source use type.

Select the desired source type from the drop-down list under SOURCE TYPE. The table will fill with the data available for that source use type. Each source use type will likely have a

different set of fuels and engines available (i.e., motorcycles and refuse trucks use different fuels and engine technologies) and, thus, will have a different set of columns displayed.

Each column represents a combination of a fuel and engine type or a summary of fractions when multiple combinations apply. Columns that are gray cannot be edited and represent such summaries.

Source Type: 21 Passenger Car

	Gasoline Conventional Internal Combustion	Diesel Fuel Conventional Internal Combustion	Advanced Gasoline <<	Gasoline Advanced Internal Combustion	Gasoline Conventional Internal Combustion - Hybrid - Mild	Gasoline Conventional Internal Combustion - Hybrid - Full	Gasoline Advanced Internal Combustion Hybrid Mild	Gasoline Advanced Internal Combustion Hybrid Full	Advanced Diesel >>
2000	99.70%	0.08%	0.18%	0.03%	0.08%	0.08%	0.00%	0.00%	0.00%
2001	99.70%	0.08%	0.18%	0.03%	0.08%	0.08%	0.00%	0.00%	0.00%
2002	99.51%	0.07%	0.38%	0.14%	0.12%	0.12%	0.00%	0.00%	0.00%
2003	99.10%	0.10%	0.77%	0.27%	0.25%	0.25%	0.00%	0.00%	0.00%
	96.37%	0.18%	3.41%	0.54%	1.12%	1.12%	0.00%	0.00%	0.00%

Click in the header near the >> symbols to expand these columns so their details can be seen.

Click the << symbol on the expanded columns to collapse them back to summary-only display.

Click on any non-gray number to allow you to change that number. All numbers are entered as [percentages](#), not [fractions](#), and can optionally include the % symbol. To enter the value of 25% (i.e. fraction 0.25), click and type

25 or 25%

Both are equivalent. Typing

0.25 will be interpreted as 0.25% (i.e., fraction 0.0025).

Normalizing Data

All entries in a single row must sum to exactly 100%. The “*Normalize*” button can be used to adjust all ratios so that all balance to 100% again. For instance, if on a model year line, two cells were available, entering 1% and 4% then normalizing, would change the percentages to 20% and 80% maintaining the original 1:4 ratio while still totaling 100%. The AVFT data can neither be exported nor the simulation run started until all model year rows on all source use types meet the above rule. The “*Normalize*” button is the easiest way to ensure this condition is met.

Adding Model Years

The AVFT panel is initialized with the contents of the MOVES Default database, including only the model years present in that database. Use the “*Add Model Year*” button to add more model years than those displayed.

Clicking this button will copy the data from the highest model year for all source use types to the next model year.

🔗 NOTE The AVFT strategy only works with model years from 2000 to 2050 inclusive and will generate an error message denying the action if an attempt is made to work outside of these bounds.

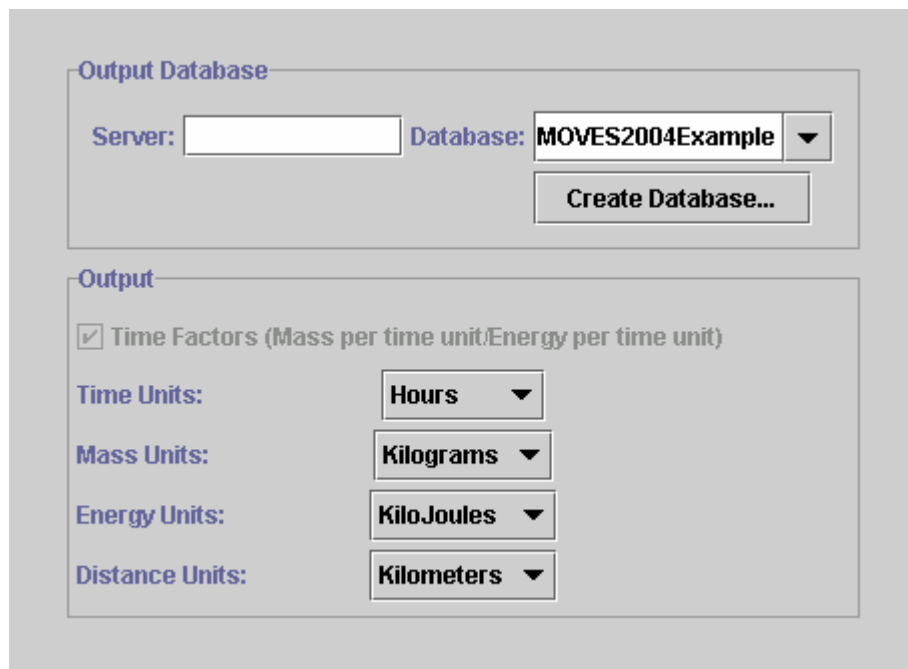
Specifying Output Dimensions

Selecting **OUTPUT** on the Navigation List expands the list to show **GENERAL OUTPUT** and **OUTPUT EMISSIONS DETAIL**. Each of these allows the user to specify specific dimensions of the output data.

Specifying Database, Time Period and Units in Output

Click on the GENERAL OUTPUT panel to specify the output database, the time period, and the units. The panel has two subpanels: OUTPUT DATABASE and OUTPUT (showing units).

CAUTION! Users must make appropriate selections in each of these sub-panels for the MOVES model to run correctly.



The screenshot displays two sub-panels for configuring output dimensions. The top sub-panel, titled "Output Database", contains a "Server:" text input field, a "Database:" dropdown menu currently showing "MOVES2004Example", and a "Create Database..." button. The bottom sub-panel, titled "Output", includes a checked checkbox for "Time Factors (Mass per time unit/Energy per time unit)" and four dropdown menus for unit selection: "Time Units" (Hours), "Mass Units" (Kilograms), "Energy Units" (KiloJoules), and "Distance Units" (Kilometers).

Choosing Output Database

The first sub-panel contains input boxes for the server name and the output database name. A blank server box is the

default and it indicates that the localhost (the MySQL database located on the user's computer) is to be used. Most users should leave this blank. The second box contains the name of the MySQL output database where the user desires the MOVES output to be written. The user must select an existing MOVES output database, or create a new MOVES output database into which the results should be placed. If the output database already exists and contains rows within its output tables, a warning icon is displayed. A new run of the MOVES model will add its results to an existing MOVES output database, and automatically give them the next available MOVES Run identification number. MOVES output records in an existing output database cannot be deleted or replaced within the MOVES GUI.

If the user chooses to create a new MOVES output database, they must choose a unique name and click the Create Database... button. Failure to click the Create Database... button will result in an incomplete MOVES RunSpec. 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 name "MOVESDEFAULT" or the name of any user supplied MOVES input databases.

Choosing Units for Time, Mass, Energy, and Distance

The second sub-panel is the OUTPUT sub-panel. It contains user choices for the time, mass, energy, and distance units in which the results will be reported. The possible choices for the time units are seconds, hours, days, weeks, months or years.

The possible choices for mass are kilograms, grams, pounds, or US tons (short tons). The possible energy units are Joules, KiloJoules, or Million BTUs (British Thermal Units). The possible distance units are miles or kilometers. The user may choose only one unit for each of the four types. The units are reported in the output database in the MySQL output table "movesrun."

The checked box TIME FACTORS (Mass per time unit / Energy per time unit) in the OUTPUT sub-panel is provided to remind the user that the MOVES output is in terms of a rate such as kilograms per hour. It also reminds the user of the interaction between the choice of engineering units in the GENERAL OUTPUT PANEL and TIME AGGREGATION LEVEL input in the TIME SPANS PANEL. By default, MOVES will set the time units in GENERAL OUTPUT PANEL to be the same as the TIME AGGREGATION LEVEL. For example, if the TIME AGGREGATION LEVEL is set at "Hour," the GENERAL OUTPUT PANEL units will be set to "Hours." The user is allowed to change the time units to the other allowable units if they choose.

The interaction between GENERAL OUTPUT PANEL time units and the TIME AGGREGATION LEVEL allows the user to report the results as a rate (mass per time or energy per time) when the Time Aggregation Level is different than the Output time units. Alternatively, the results are reported as a total inventory quantity mass of emissions or total energy over the output time period when the Time Aggregation Level is the same as the Output units. This important interaction is best illustrated by a couple of simple examples.

Example 1: The user chooses Time Aggregation Level of HOUR and Time Units of SECONDS and Mass Units of Kilograms.

The output record in Example 1 will reflect the rate of emissions during one hour, expressed as a rate of kilograms per second. It is computed internally in the model by dividing the quantity of emissions in the one hour (in kilograms) by 3600 seconds to obtain the rate in Kilograms per second.

Example 2: The user chooses Time Aggregation Level of HOUR and Time Units of HOURS and Mass Units of Kilograms.

The output record in Example 2 will reflect the emissions rate during the hour, expressed as a rate of kilograms per hour. Because the Time Aggregation Level and the Time Units are the same (Hours), they effectively cancel and the result is the total emissions in Kilograms in the hour.

!TIP As a result of the interaction between the TIME AGGREGATION LEVEL and the OUTPUT time units, it is suggested that the user select OUTPUT time units that are equal or shorter than the selected TIME AGGREGATION LEVEL. This will avoid potentially misleading results such as reporting an Hour's quantity of emissions with a rate such as kilograms per month.

Specifying Emission Distinctions in Output

Select **OUTPUT EMISSIONS DETAIL** on the Navigation List (after expanding Output) to specify distinctions desired in the output data. The OUTPUT EMISSIONS DETAIL panel consists of four subpanels: ALWAYS, FOR ALL VEHICLE/EQUIPMENT CATEGORIES, ESTIMATE UNCERTAINTY, and ON ROAD/OFF ROAD.

The screenshot displays the 'OUTPUT EMISSIONS DETAIL' panel with the following subpanels and options:

- Always**
 - Time (Hour dropdown)
 - Location (COUNTY dropdown)
 - Pollutant
- for All Vehicle/Equipment Categories**
 - Model Year
 - Fuel Type
 - Emission Process
- Estimate Uncertainty (grayed out)
- On Road/Off Road**
 - On Road/Off Road
 - On Road**
 - Road Type
 - Source Use Type
 - SCC
 - Off Road**
 - Segment
 - SCC
 - HP Class

The more distinctions made, the more records the output database will contain.

!TIP If the scope of a run is large (e.g., includes many times, location, sourcetypes, etc.), then the output must be highly aggregated (i.e., have few items selected on this screen) or its size will be unmanageable.

🔗 NOTE The option to "Estimate Uncertainty" has no effect in this version of the model, so is grayed out.

Always Box

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," "Day," "Month,"* or *"Year."*

By default, the *"Location"* button will report the same location level as selected in the MACROSCALE GEOGRAPHIC BOUNDS panel. The user may change the location level in the *"Location"* button to a value that is equal or larger than the level set in the MACROSCALE GEOGRAPHIC BOUNDS panel. For example, if the MACROSCALE GEOGRAPHIC BOUNDS panel is set to *"County"*, then the *"Location"* button can be set to *"County," "State,"* or *"Nation"*.

No *"Pollutant"* button is available because the pollutants are always reported separately in the output (there is no aggregation across pollutants).

For All Vehicle/Equipment Categories Box

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 *“Model Year”* is checked, then the output will be broken down by Model Year.

On Road/Off Road Box

The right hand side of this screen reflects the fact that MOVES2004 includes only On Road Vehicles. (The ON ROAD section is always available and the OFF ROAD section is always grayed out in MOVES2004.)

Within the ON ROAD section, options are available to report by *“Road Type,” “Source Use Type,”* or *“SCC”*. If none of these options are checked, then no vehicle classification distinctions are made in the output. Instead, the results are reported summed across all vehicle classes.

!TIP *“Source Use Type”* and *“SCC”* are mutually exclusive.

Source UseTypes are native to MOVES2004, so this is the preferred option. MOVES2004 can report by SCC, but this introduces an additional approximation step in the calculations to convert output by SourceUseType to SCC.

Several interdependencies exist among these items. For example, reporting by SCC implies that RoadTypes and FuelTypes will be distinguished.

3. Executing Example MOVES2004 Run Specification

The example MOVES2004 Run Specification models the entire U.S for one year (the base year 1999) at the most aggregate level. It estimates total energy consumption and emissions of methane and nitrous oxide for the running, start, extended idle and well-to-pump processes, for all vehicle (use) types (gasoline, diesel, and CNG transit buses) and roadway types. Vehicle Miles Traveled (VMT) is also estimated.

The example run specification is located at the top level MOVES directory. Its name is MOVES2004Example.mrs. (The .mrs extension, for "MOVES Run Specification" is suggested but not required. Run specification files are XML-formatted text.)

Assuming that MySQL and MOVES2004 have been installed, that the MySQL server and the MOVES MasterGUI program have been started, and that MOVES Worker program(s) are operational, the example Run Specification can be executed by:

1. Selecting File, then Open (on main Menu Bar).
2. Navigating, if necessary, to the top level MOVES directory.
3. Selecting the MOVESEExample.mrs file.
4. Select General Output (on Navigation List down left side panel). Select or create the output database (provide a database name) into which the results should be placed.

5. Select Action, then Execute (on main Menu Bar). This run specification takes approximately 10-12 minutes to execute on a single computer configuration, using a late model PC.
6. The output will appear in a MySQL relational database. The name of the database will be the same one specified Step 4. The database will typically contain four tables. These are:
 - MovesOutput - This table contains the Emission results of the run disaggregated by several operating parameters such as Year, Month, Day, Source Type, County, Fuel, Model-Year, Road-type, SCC, Emission Process, etc.
 - MovesActivityOutput - This table contains the VMT results of the run disaggregated by several operating parameters such as Year, Month, Day, County, Fuel, Model-Year, Road-type, SCC, etc.
 - MovesRun - This table contains 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.).
 - MovesError - This table contains any error messages or diagnostics that might occur if the MOVES run is unsuccessful.

4. Customizing MOVES 2004

Although a full range of default data is available in MOVES2004, the model was designed to give user the ability to customize the model to meet their specific needs. The primary reasons users would customize MOVES are:

- to better represent a local area
- to alter inputs for “what-if” analysis, such as the use of different advanced technology or fuel scenarios.

Local Customization

The default modeling domain for MOVES2004 is the entire U.S. plus territories, with counties defined as zones. To customize MOVES2004 for a specific area, first redefine the modeling domain to which the input data pertains, as well as the zones that make up the domain, if sub-domain analysis was desired. The minimum level of data required in tailoring input data to redefine domain would be VMT and geographic (zone-level) allocation factors, since the current defaults only apply to the entire nation. All other fleet and activity inputs only serve to allocate these total activity measures; hence, if not changed, the national default allocations by time, roadway type, age, source type, etc. would be applied to the user-supplied VMT and geographic allocation data.

Beyond this basic requirement, local customization could proceed on many levels. On the simpler end, it may involve using local vehicle age distribution or meteorology data, similar to what many users currently undertake with MOBILE. On the

more complex end, the level of local customization could extend to specific driving schedules culled from in-use driving surveys.

The more specific the data for an area, the better the output will be for that area. Given the array of choices a user faces in using local data, a sensitivity analysis will be performed on MOVES2004 to shed light on the most important factors to focus on when customizing.

“What-if” Analysis Customization

MOVES2004, with integrated GREET, will provide a very broad range of “what-if” analysis capability for looking at different future scenarios. “What-if” capability extends to four dimensions: fuel and vehicle technology penetration, energy and emission performance of these technologies, upstream fuel pathway options, and activity patterns. How to make changes in each of these dimensions follows.

Fuel and Vehicle Technology Penetration

As discussed in Section 2, the ALTERNATIVE VEHICLE FUELS AND TECHNOLOGIES (AVFT) STRATEGY Panel has been developed to provide a convenient graphically-driven mechanism for the user to input different penetration rates of the broad range of vehicle and fuel combinations available in MOVES, by source type and model year. This allows the user to address the question “What is the impact of having X percent of advanced technology Y in the fleet, in year Z?” The control strategy is designed to make the necessary changes to the underlying MOVES database tables

feeding into the Source Bin Generator (See the MOVES2004 Software Design Reference Manual).

Energy and Emission Performance

The external control strategy Future Emission Rate Creator (FERC), discussed in Section 2, has been developed to allow the user to input alternative assumptions regarding the relative benefit of energy and emission performance of individual technologies. This allows the user to address the question “What is the impact of varying the energy consumption and emissions of advanced technology Y?” (See Section Main Menu / PreProcessing / UPDATE FUTURE EMISSION RATES for more details on how to generate and add Future Energy Consumption and Emission rates to MOVES2004).

Fuel Pathway Options

The integration with the Argonne National Laboratory well-to-wheel model GREET will enable the user to select different pathway options, or mixes of options, for specific fuel subtypes in MOVES. It will also allow the user to alter inputs at each step of a given pathway. This allows the user to address the question “What is the impact of varying the fuel pathway specifications for the fuel used in advanced technology Y?”

Activity Patterns

MOVES2004 provides unprecedented flexibility in varying activity patterns of the fleet, for more real-world estimation. This pertains to many parameters related to VMT growth and allocation, as well as to driving patterns themselves. Specifically, MOVES2004 estimates energy and emissions using

real-world driving patterns culled from in-use driving surveys. The user can look at differences between real-world urban vs. freeway driving or congested vs. uncongested conditions, across source use types. The user would alter such inputs directly in the underlying MySQL database, through use of the "Manage Input Data Sets" screen on the MOVES2004 graphical user interface. The intent for future versions of MOVES is to develop an internal control strategy, or perhaps several, to address alternate activity inputs.

Appendix A. Using GREET to update the Well-to-Pump Emission Factors

A version of GREET is included in the MOVES Installation which can be used in conjunction with MOVES2004 to update the emission factors MOVES uses to calculate results for the well-to-pump emission process. In MOVES2004 these factors are a function of calendar year, pollutant, and fuel subtype. In GREET they are a function of many factors.

This section provides a brief overview of the GREET model within the MOVES model framework. The document "User Manual and Technical Issues of GREET for MOVES Integration" provides more details on the operation of the GREET model.

⚠CAUTION! GREET is not open source software and requires that Microsoft Excel be installed on the computer running the MOVES GUI program for this option to be used.

If Excel is available, GREET can be invoked from MOVES2004 to update these rates as follows:

1. Close any open Excel spreadsheets.
2. Load or create a run specification containing the calendar year(s) and fuel types (selected on the "On Road Vehicle Equipment" screen) for which updated well-to-pump rates are desired.

⚠CAUTION! An error message will appear if this step is not done before executing GREET.

3. Select UPDATE WELL-TO-PUMP RATES... under PRE-PROCESSING on Menu Bar.
4. When prompted, specify the name of the input database where GREET results should be placed. New database names are OK.
!TIP the user should choose a database name other than MOVESDefault, so as not to overwrite the default GREET inputs in MOVESDefault. This will also allow an alternate GREET run to be applied to several RunSpecs.
5. Accept the default "OK" response to the warning about needing to close any open Excel spreadsheets.
6. After a brief pause, an introductory screen to GREET GUI version 1.6 should appear. Select "New Session."
7. GREET GUI will display a series of screens which can be used to accept or alter various input GREET input data.
8. At the end of this process GREET GUI will indicate it is executing.
9. Exit the GREET GUI session and reopen the MOVES GUI Window
10. MOVES will indicate that the Well-to-Pump Factors have been updated. An alternate database table "GreetWellToPump" will be placed in the user-specified input database for MOVES to overlay on the default database. This database will be available for future runs as well or for use with alternate Runspecs.

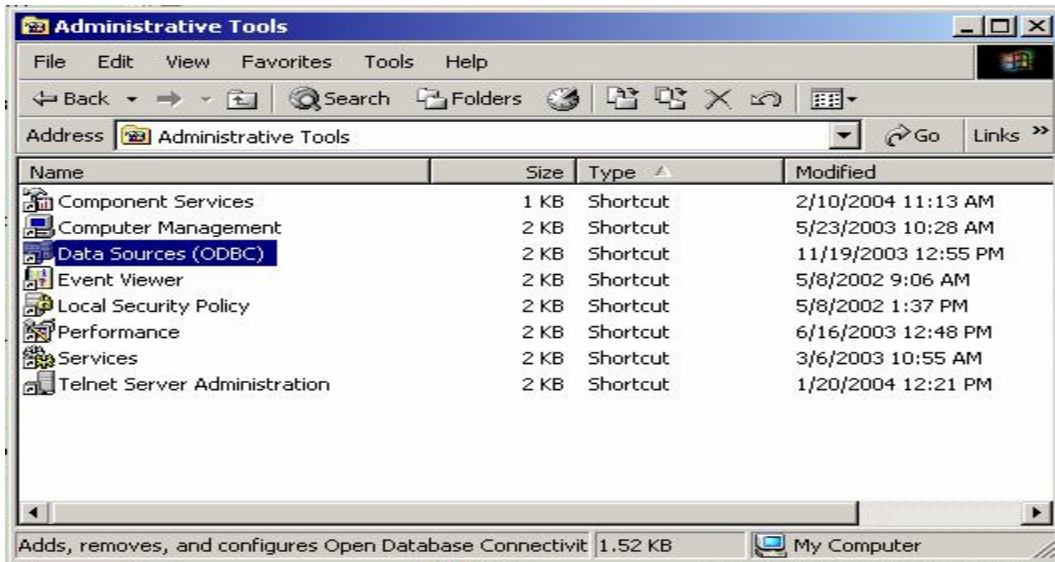
Appendix B. Linking MySQL tables from MS ACCESS (includes ODBC usage)

This Appendix describes in detail how the user can transform the MOVES MySQL output tables into Microsoft ACCESS database tables or Microsoft Excel spreadsheets. This may be useful for users who do not wish to access MOVES input and output databases in MySQL format. Appendix B also explains how to use the Open Database Connectivity interface protocols and tools.

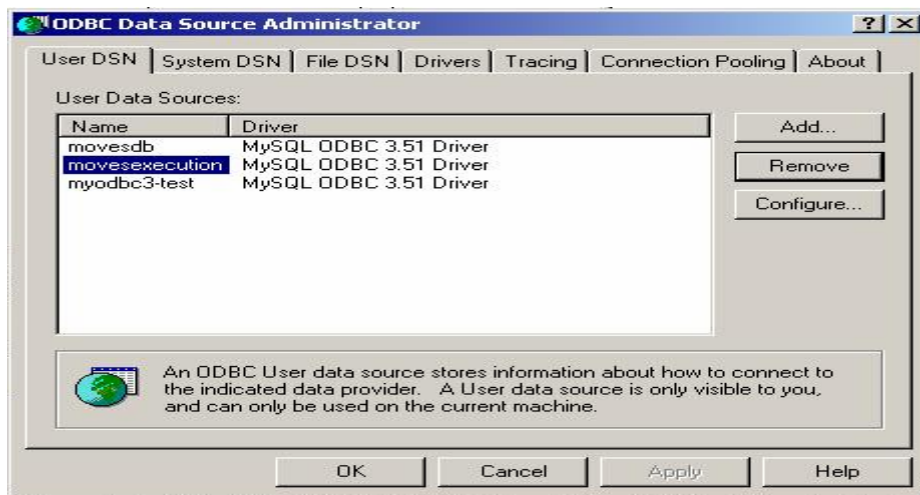
1. Install MySQL driver, MyODBC-3.51.06.exe.
2. Create an ODBC data source named the same as the folder which corresponds to the MySQL database.
 - (i.) In the control panel select administrative tools :



(ii.) In the administrative tools select Data Sources (ODBC) :



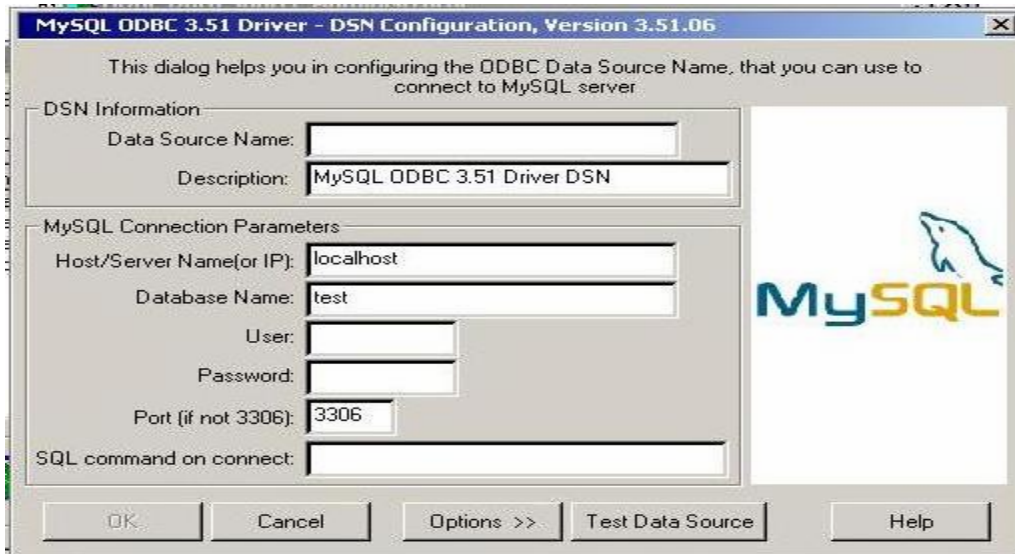
(iii.) In the ODBC Data Source Administrator dialog choose to “Add” a “User Data Source”



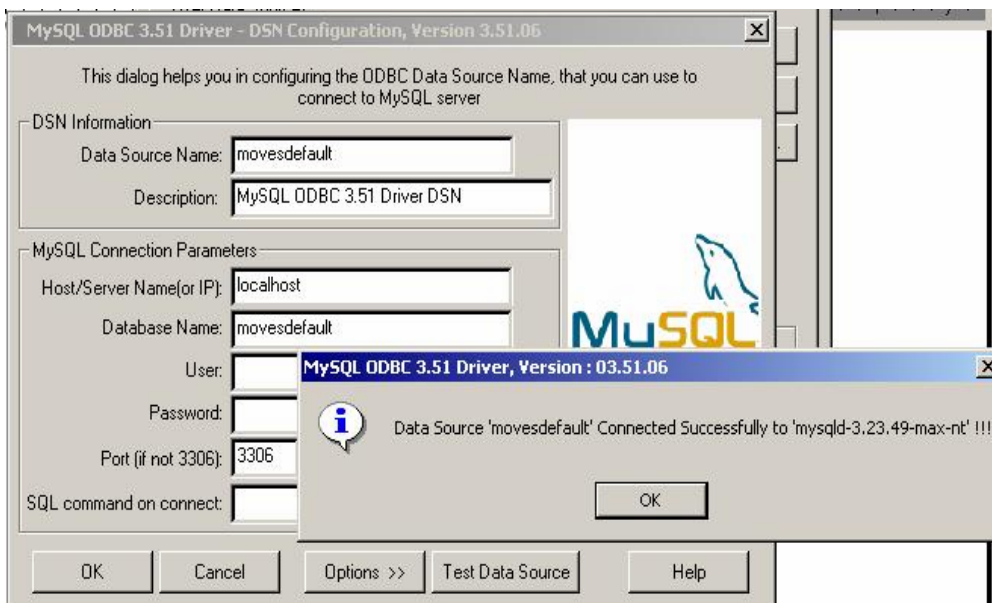
- (iv.) The “New Data Source” should use the MySQL ODBC Driver that was installed earlier. So, scroll through the list until the MySQL ODBC driver is found, highlight it, and click the “Finish” button.



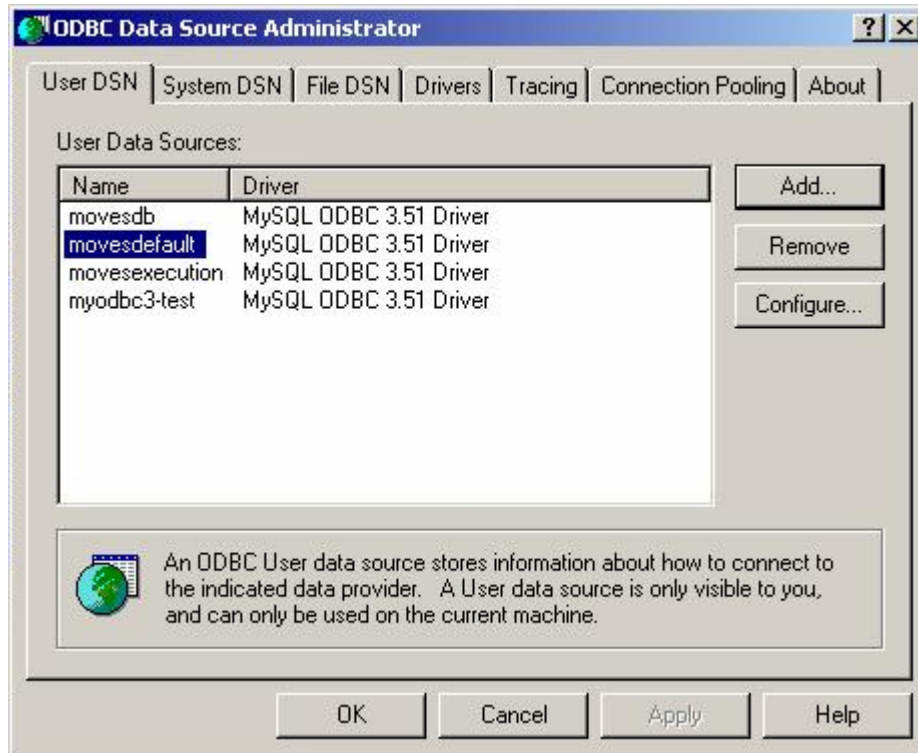
- (v.) Next, the database links need to be defined. The default values are as shown below. The database name will need to be changed to the name of the particular folder which contains the MySQL database files (i.e., the MySQL database name).



- (vi.) Once the database name and the data source name has been changed to an existing MySQL database, the database connection can be checked by clicking on the “Test Data Source” button.

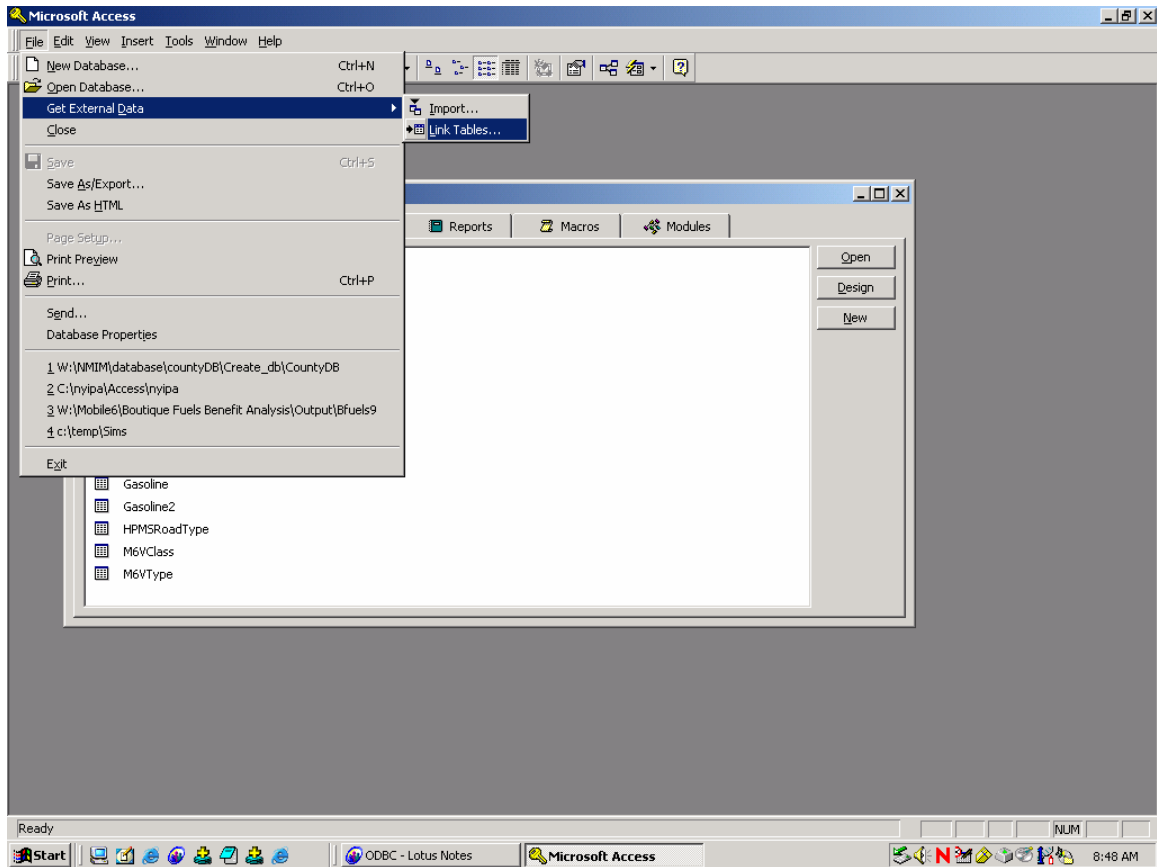


- (vii.) The added data source will show up in the list of data sources in the ODBC Data Source Administrator dialog. And this will finish the process of creating a new ODBC data source associated with a particular MySQL database.

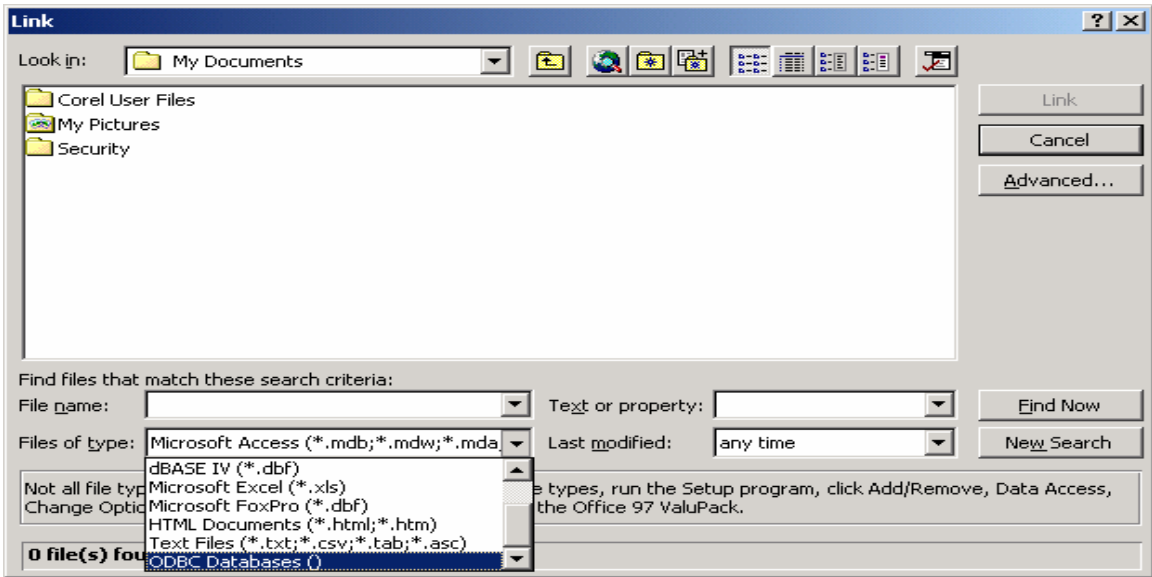


3. Run an ACCESS .mdb (either a new one or an existing one)
4. On the top menu bar, look for menu items, “File”, “Get External Data”, then “Link Tables ...” (see diagram below). Click on the item “Link Tables...”

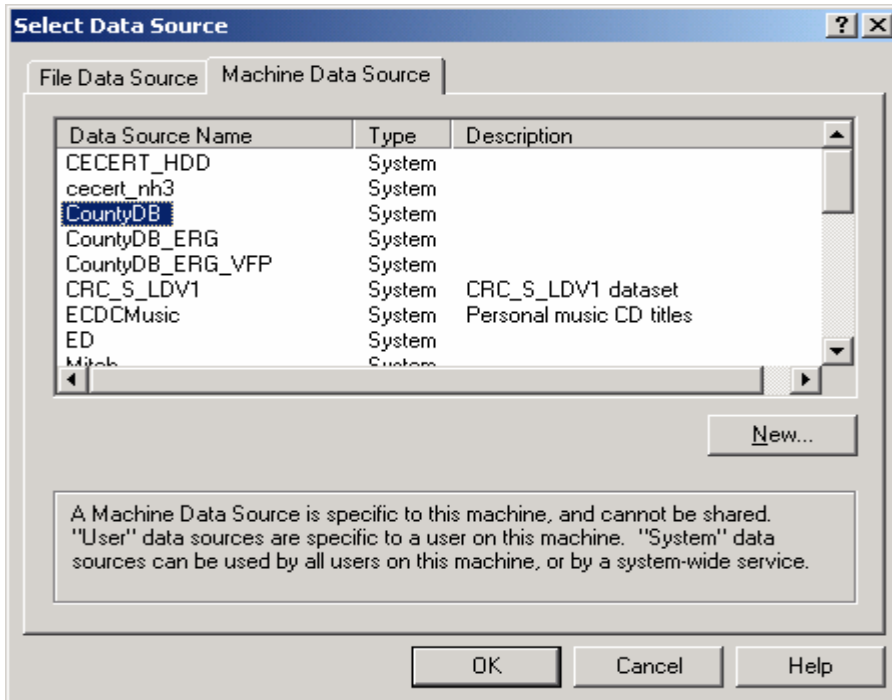
Note: Using "Import" instead of "Link Tables" in Microsoft Access 97: There's a known problem with Microsoft Access 97 when linking MySQL tables that have any fields that are BIGINT data type, e.g., the BIGINT field SourceBinID in tables EmissionRate, SourceBin, and SourceBinDistribution of the MOVES database. The Microsoft Access 97 does not recognize the BIGINT data type and consequently will fail to link these tables. It is recommended to use the Microsoft Access "Import" feature to retrieve data from MySQL databases. There're many get-arounds to overcome this problem, however using "Import" feature is the simplest one to use.



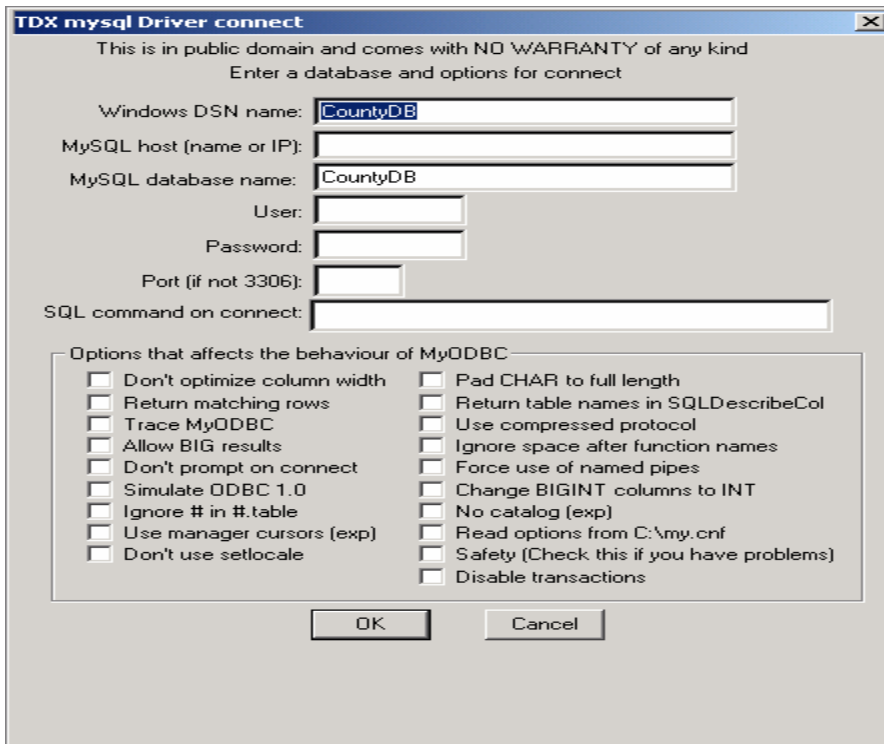
4. On the pop-up windows like the one below. Choose “ODBC Databases()” from the drop-down list which is located next to the label “Files of type” (you may need to scroll down a bit from this drop-down list box to find “ODBC Databases()”).



5. Another pop-up window similar to the one below will show up. Click on the tab “Machine Data Source”; select “CountDB” from “Data Source Name” column. (if you can not find “CountDB” under “Data Source Name” column, try to find it under tab “File Data Source”. If you still can not find it, let me know so I can come over to help).

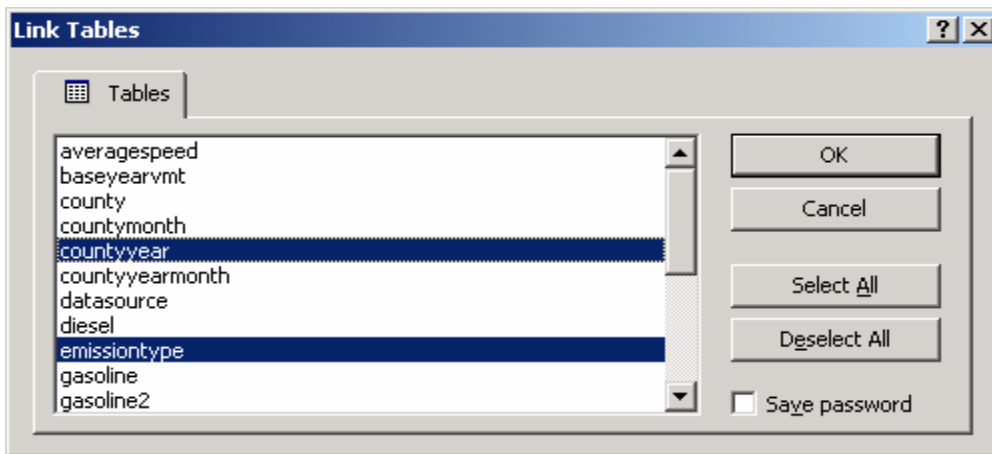


Click on “OK” button to invoke the ODBC window,

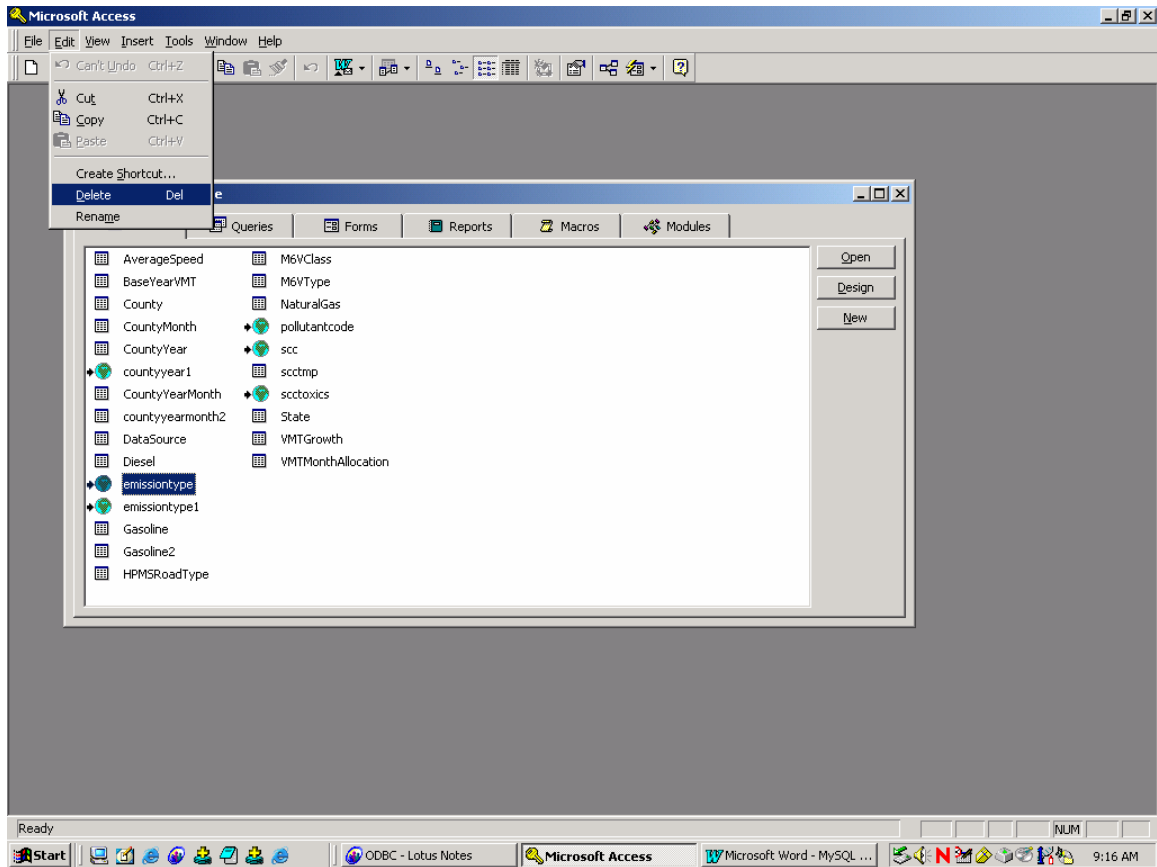


Do nothing , just click on “OK” button.

6. On the “Link Tables” window as shown below, select as many tables as you want, click on “OK” button. You should see the linked tables in your ACCESS database screen under tab “Tables”.



7. To remove the link(s) to MySQL table(s), select the table(s) in ACCESS mdb, then select “Delete” under “Edit” on top menu bar (see picture below). This means only the link, not the MySQL table(s) itself, will be deleted from your ACCESS mdb. You can always re-link any MySQL tables later.

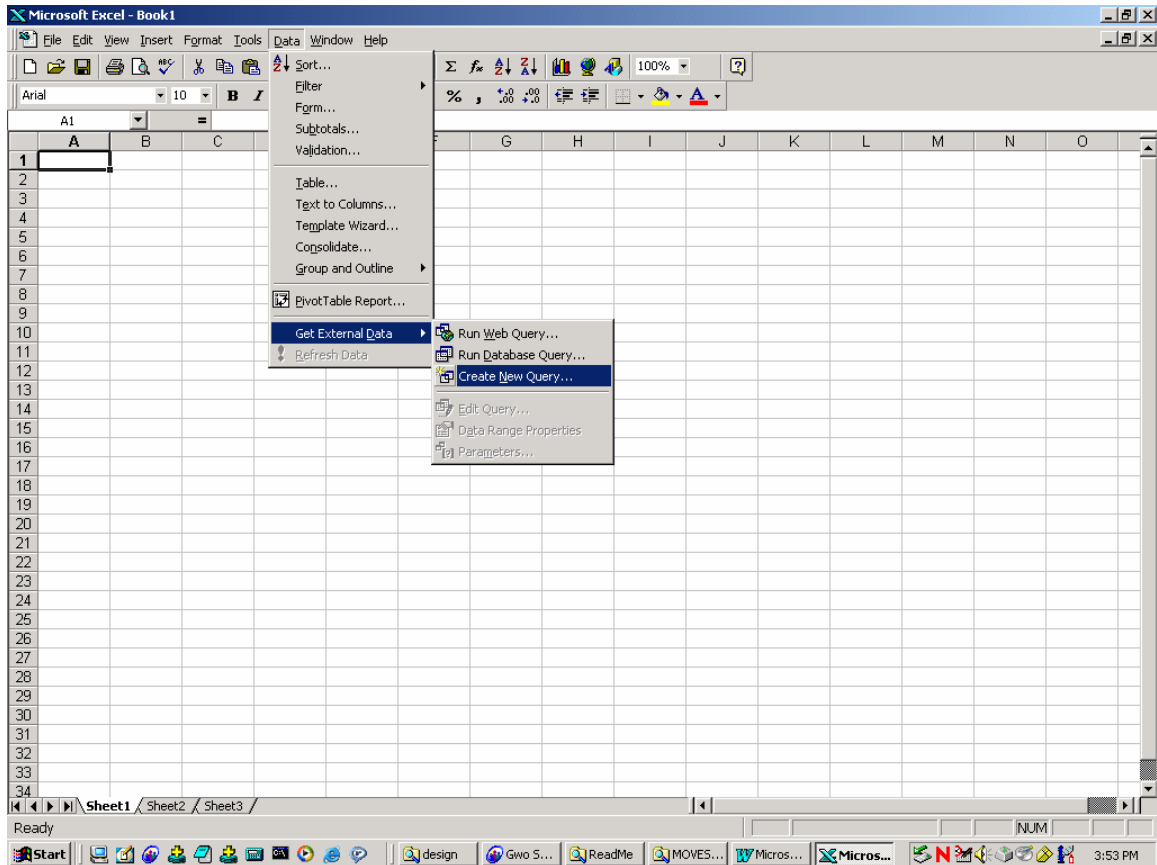


Accessing MOVES MySQL databases using Microsoft Excel 97

3. Make sure the MySQL ODBC driver 3.51 (see the document “MySQL from an ACCESS mdb.doc”) or higher version has been installed on your computer. If not, referring to the MOVES installation package for the install program MyODBC-3.51.06.exe and its documentation.

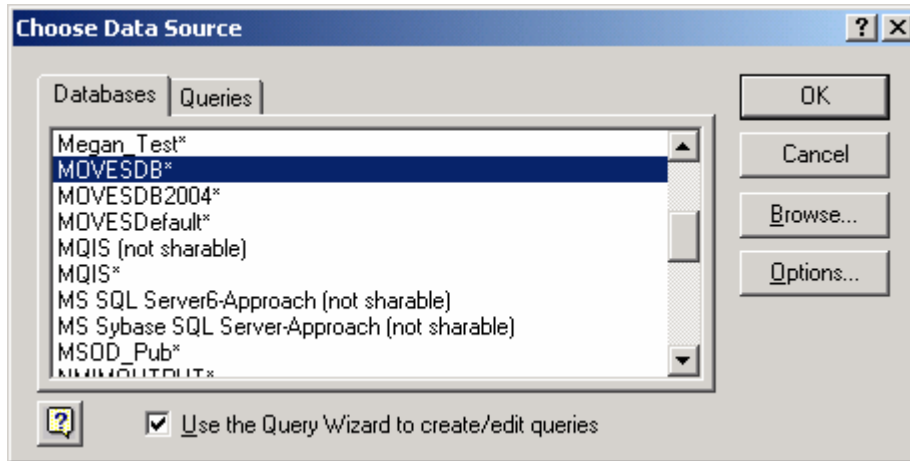
4. Make sure the ODBC data source name(s) (DSN) pointing to the desired MySQL database(s) have been created on your computer. If not, referring to the step 2 of the Section “Linking MySQL tables from Microsoft ACCESS 97”. Note that if a DSN has been created when

setting up for Microsoft Access, that DSN can be shared by Excel and vice versa.

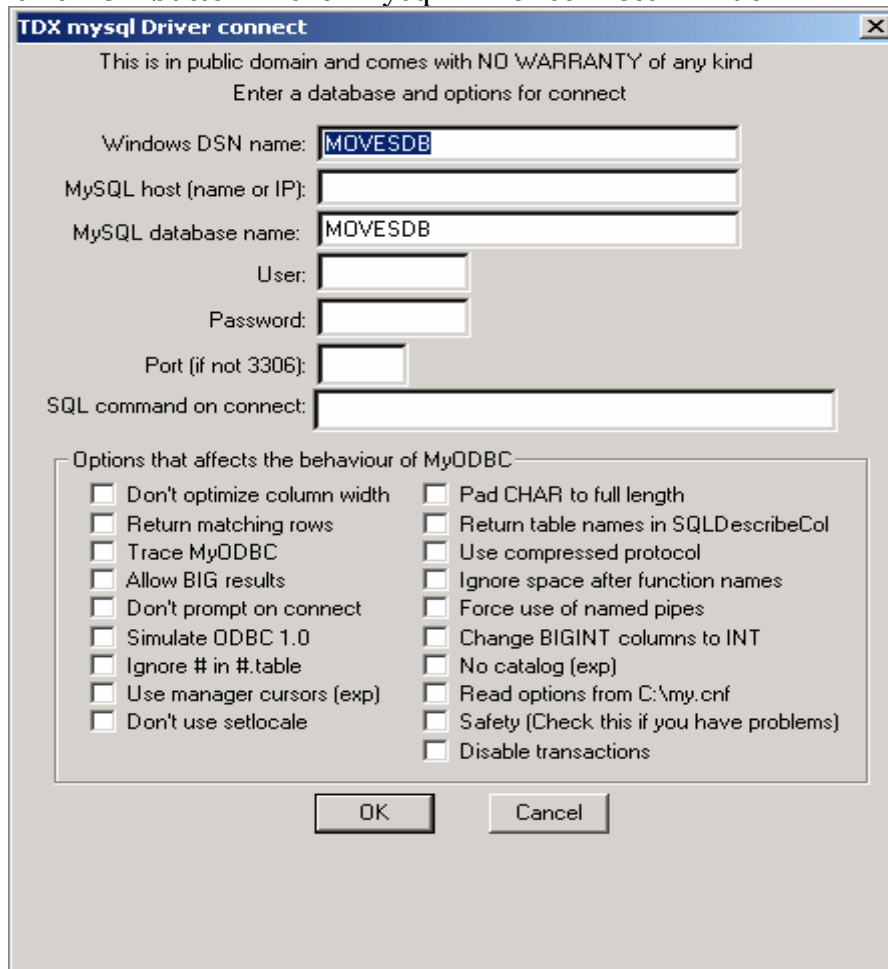


5. Open an Excel file, click on Data from the top menu bar, click on Get External Data from the popup menu, then select “Create New Query ...” from the sub-menu window (see diagram below). This will bring up a window of “Choose Data Source” window.
6. Assuming that the ODBC data source name(s) (DSN) pointing to the desired MySQL database(s) have been created on your computer as described in the Item 2 above. From the Choose Data Source window,

select the desired DSN from the list, e.g., “MOVESDB”, then click on OK button.



7. Click on OK button in the “mysql Driver connect” window



8. The Query Wizard shows up, follow the instructions posted by the Wizard until finished.

