MOVES-HVI
Demonstration Version
User Guide

Draft

MOVES-HVI
Demonstration Version
User Guide

Draft

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

Based on the MOVES 2004 User Guide
Prepared by the MOVES Development Team

Updated for MOVES-HVI
by Mitch Cumberworth
with assistance from Megan Beardsley
Ed Glover
and John Koupal

Table of Contents

1. Getting Started

The Highway Vehicle Implementation of the MOtor Vehicle Emission Simulator (MOVES-HVI) is EPA's second implementation of MOVES. (The first implementation was called MOVES2004.) 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 MOVES-HVI

MOVES-HVI retains most of the functionality of MOVES2004 and can be used to estimate national inventories and projections at the county-level for energy consumption (total, petroleum-based and fossil-based), carbon dioxide (CO_2), nitrous oxide (N_2O), and methane (CH_4) from highway vehicles. While MOVES-HVI can include life cycle (i.e., well-to-pump) effects in the estimate of energy consumption and emissions, it no longer includes an interface to the GREET model, produced by Argonne Laboratories, which could be used in MOVES2004 to adapt well-to-pump emission rates to a variety of assumptions.

MOVES-HVI adds significant functionality to MOVES2004 to estimate the criteria pollutant emissions of highway vehicles (HC, CO, NOx, and PM). This version of MOVES-HVI is intended for demonstration purposes only. None of its numerical results should be considered accurate or realistic. 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, and operate at smaller scales.

MOVES 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, which covers 3222 counties of the United States and which supports model runs for calendar years 1990, and 1999 – 2050, is included with the model.

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

About MOVES-HVI Documentation

The following discussion highlights what is available in this manual, how to best use it, and where additional information may be found.

About this Manual

This MOVES-HVI User Guide focuses on operating MOVES software to create and execute run specifications (RunSpecs).

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".

This release of MOVES is intended to work only in a Microsoft Windows 2000 or later Windows environment (i.e., Windows Versions 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.

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 MOVES-HVI:

- **Installation:** A README text file 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-HVI Software Design and 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. It is intended to answer more detailed and substantive questions about the MOVES software.
- **MOVES Database Documentation:** More detailed documentation of the MOVES database can be found within the readme file in the database itself.

The MOVES-HVI User Guide and the MOVES-HVI Software Design and Reference Manual are available as .pdf files from within MOVES-HVI itself. These are 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 **Software Design and 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

Step-by-step instructions on installation are supplied by EPA in the README file in the MOVES-HVI Software Distribution Suite. (A somewhat more detailed discussion is also available in the MOVES Software Design and Reference Manual.) The MOVES-HVI Software Distribution Suite consists of three required installation programs (MySQL, Java1.4 and MOVES-HVI

itself), and several additional programs that can be used to install additional, optional software tools for use with MOVES. Please refer to those materials first if MOVES-HVI has not been installed on your computer.

System Requirements

The MOVES program is open source and written in JAVA and MySQL, but has currently been developed 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 256 MB of RAM, (512MB 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 700 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. Extensive users of MOVES 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 configurations.

Testing Your Installation

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

1. Check that the MySQL server is operating on all computer systems in the 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 MOVES 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.

NOTE This step is not necessary if operating with a single computer configuration.

**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.

2. Using MOVES-HVI

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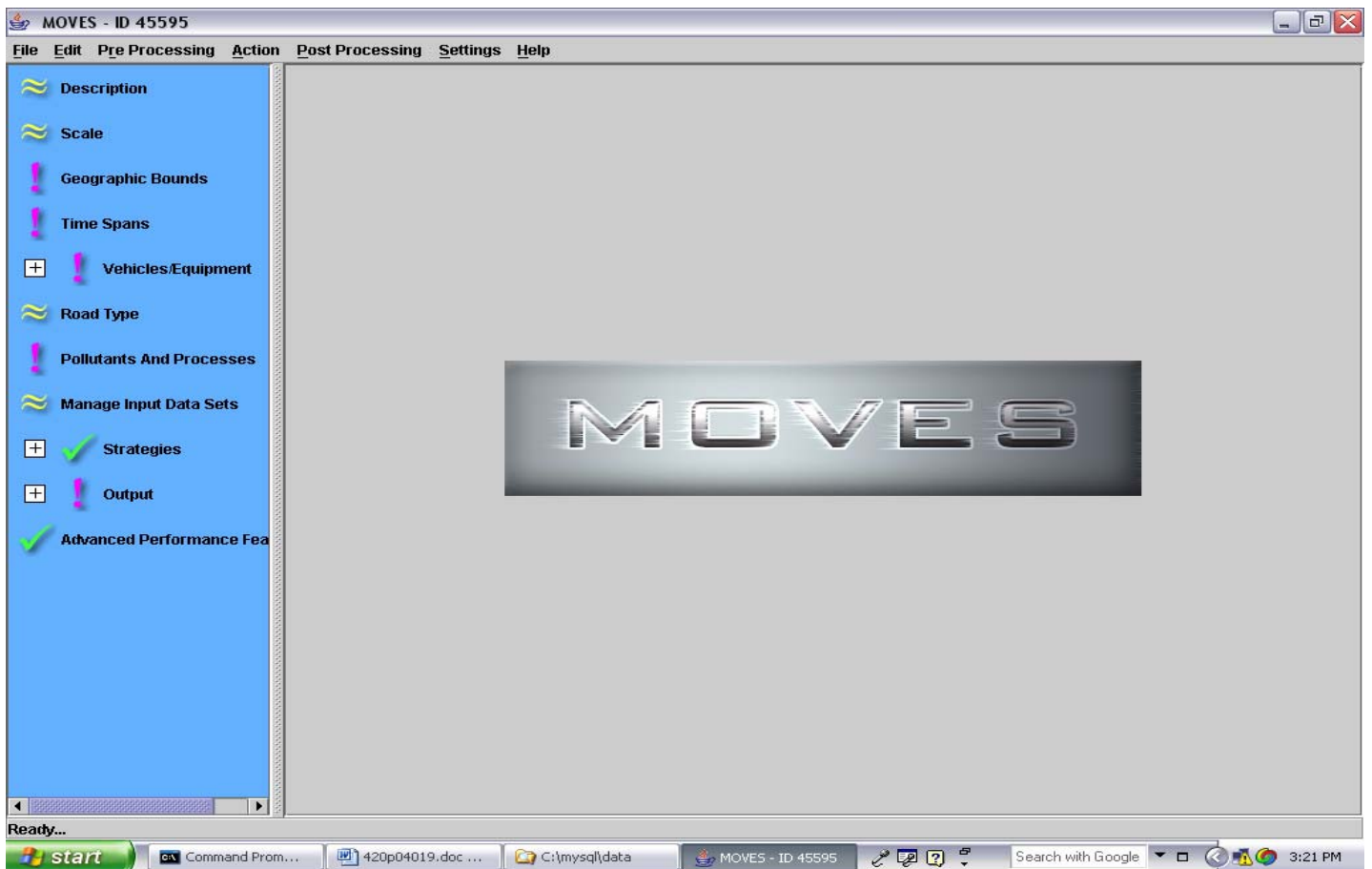
The MOVES RunSpec Graphical User Interface (GUI) consists of three parts: Main Menu Bar, Navigation List, and Detail Panel. The Menu Bar and Navigation List show on every screen with the Detail Panel changing as items on the Navigation List are selected.

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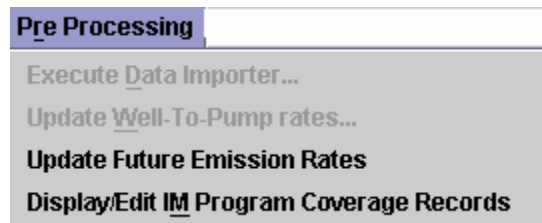
File provides a drop-down menu typical of those used when manipulating documents. Drag and click to select New, Open, Close, Save, Save As, Print, previously opened files, and Exit. Use these to create, load from disk, and save RunSpec objects. The Example
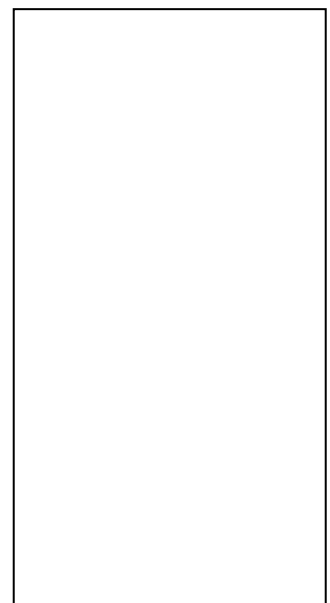
RunSpec may be opened from this menu (see Section 3 for details).

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

PRE PROCESSING contains four items, of which only two are operable in the current version.

| Pre Processing |
|---|
| Execute Data Importer... |
| Update Well-To-Pump rates... |
| **Update Future Emission Rates** |
| **Display/Edit IM Program Coverage Records** |

The UPDATE FUTURE EMISSION RATES menu item allows the user to create energy, CH4 and N2O 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** The default database provided with MOVES-HVI provides default rates for alternative fuel and advanced

technology vehicles for model years 2001-2010, and for any post-2010 model year vehicle. The user is therefore not required to generate these rates using the FERC unless 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.

Future Emission Rate Calculation in MOVES

Future Emission Rate Database
Server: localhost
Database: FERCInputDatabase

Execute
Cancel

Short-Term Future Emission Rate File
C:\EPA\MOVES\FERC\ShortTermFERAdjustment.csv   Browse...

Long-Term Future Emission Rate File
C:\EPA\MOVES\FERC\LongTermFERAdjustment.csv   Browse...
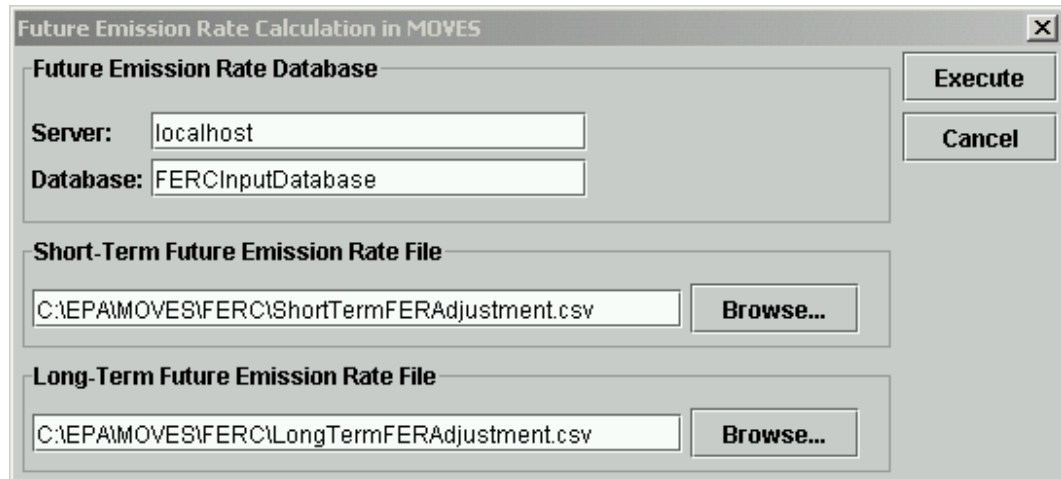
♀ **NOTE** The user cannot name the database containing the
future emission rates "MOVESDefault" or
"MOVESExecution". It is also highly recommended that
the server name remain "localhost".

Examples of the "Short-Term Future Emission Rate File"
and "Long-Term Future Emission Rate File" can be
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, 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, using the description in Section 10.36 of the
"MOVES-HVI Software Design Reference Manual".

After the appropriate inputs are made to 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. Selecting 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.

The DISLAY/EDIT IM PROGRAM COVERAGE RECORDS... menu option allows the user to display and change the information about vehicle inspection maintenance programs that applies to the current run specification. Because using this feature requires an understanding of other parts of the run specification it is covered later in this document in Appendix A.

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
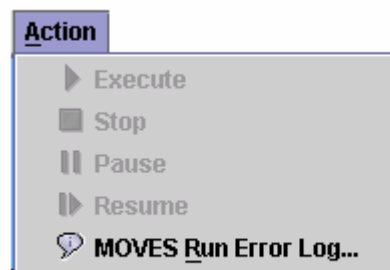
the required RunSpec inputs have been satisfied. Until they are satisfied, the EXECUTE item will remain grayed out. After the user satisfies the RunSpec input requirements (all check marks in the navigation panel must be set to green checks or yellow tildes – see

"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.
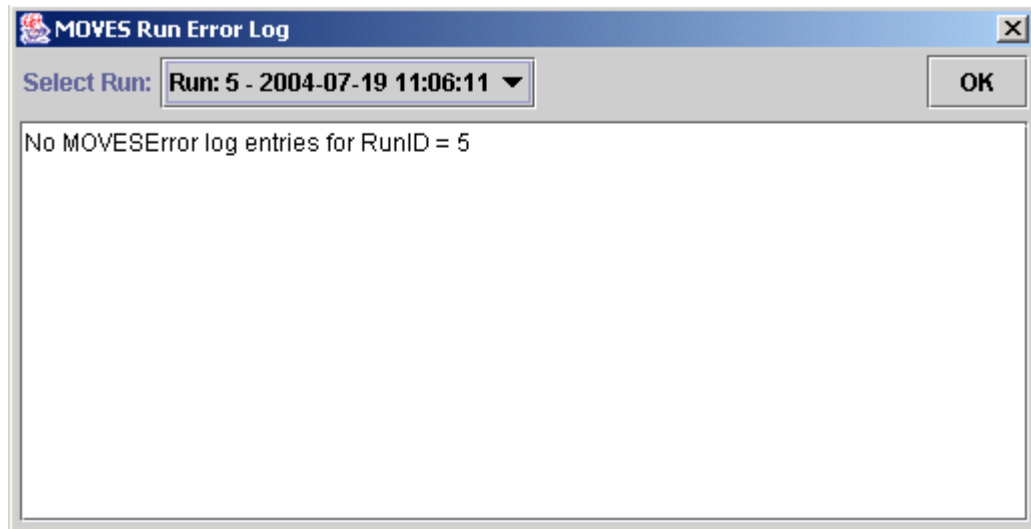
ꝋ **NOTE** A program that has been stopped cannot be resumed.

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

ꝋ **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.

NOTE Not all error messages necessarily mean that the run results are incomplete or invalid. These messages are generally worded as "Warning" messages, e.g. "Warning: RunSpec doesn't have all the RoadTypes".

MOVES Run Error Log

Select Run: Run: 5 - 2004-07-19 11:06:11 ▼   OK

No MOVESError log entries for RunID = 5

POST PROCESSING has two items, "Run MySQL Script on Output Database" and "Produce Summary Report". These features can be used after a successful execution of MOVES to further process the MOVES results into more aggregate or easy to use forms.

Post Processing

Run MySQL Script on Output Database
Produce Summary Report

The Run MySQL Script on Output Database item consists of a set of MySQL Scripts that process the MOVES output databases (stored in the MySQL database format).

Click "Run MySQL Script on Output Database" to open the SELECT SCRIPT box, as shown here.
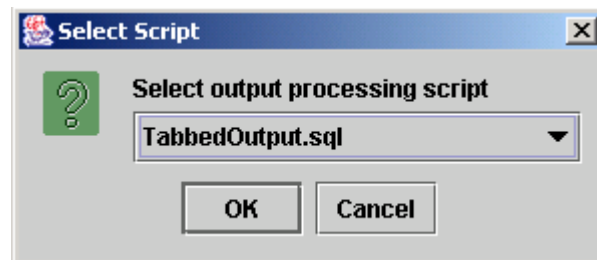
Select the "OK" button to <u>execute</u> the script shown in the center scroll down list. In this example, it is the "TabbedOutput.sql" script.

Click on the arrow in the scroll down box to view all the available scripts, as shown here.
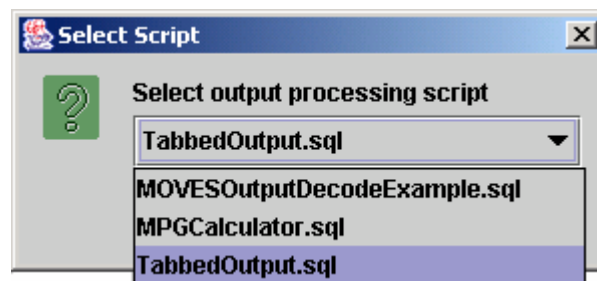
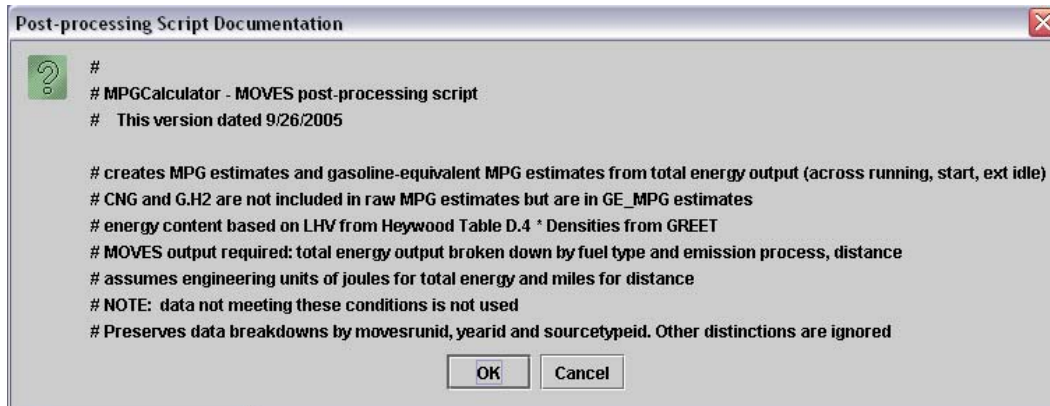Select a post-processing script by clicking to highlight it.

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 and that certain engineering units have been used in the MOVES run.   Otherwise, the results will be meaningless.

The third script "TabbedOutput.sql" takes 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. In MOVES-HVI this can also be done with the Summary Reporting feature.

When a script is selected a pop-up window is displayed giving the user information about the script. This may explain some limitations of the script. For example, the following popup is displayed by the MPGCalculator script:

Post-processing Script Documentation

```
#
# MPGCalculator - MOVES post-processing script
#   This version dated 9/26/2005

# creates MPG estimates and gasoline-equivalent MPG estimates from total energy output (across running, start, ext idle)
# CNG and G.H2 are not included in raw MPG estimates but are in GE_MPG estimates
# energy content based on LHV from Heywood Table D.4 * Densities from GREET
# MOVES output required: total energy output broken down by fuel type and emission process, distance
# assumes engineering units of joules for total energy and miles for distance
# NOTE:  data not meeting these conditions is not used
# Preserves data breakdowns by movesrunid, yearid and sourcetypeid. Other distinctions are ignored
```

OK   Cancel

The user may Cancel execution of the script at this point or execute it by selecting "OK".

!TIP   The Post Processing feature of MOVES is also allows the user to run user-created 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 Structured 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, which is named "MOVES", in the subfolder named "Database", in a subfolder named "OutputProcessingScripts".

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

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

If the script contains an initial block of comments this will be displayed to the user before the script is actually executed.

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

The **Summary Report** item of the Post-Processing Menu can be used to summarize and report the output of one or more MOVES runs in a variety of ways. These reports can be viewed on the screen, printed, or saved in an ASCII text form that is easily imported into other software such as Microsoft Excel. The Summary Reporter is an important feature added to the MOVES-HVI implementation of MOVES. Its usage is covered in Appendix C.

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.
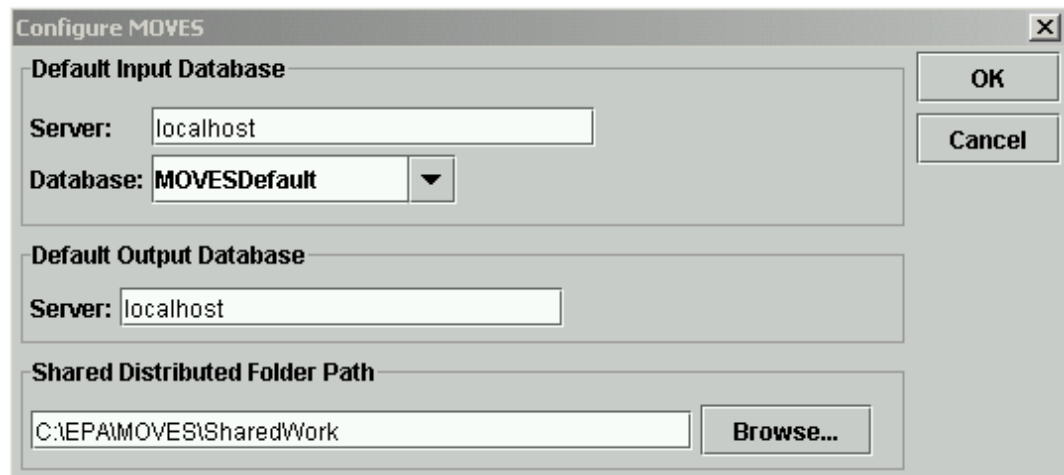
✐**CAUTION!** If the Default Input Database is changed the program should be stopped and restarted. This is because the UI program has already connected to the previously specified database in order to have constructed the GUI displays.

HELP provides access to the online MOVES User's Guide and Software Design and Reference Manual and to some general information about the version of MOVES you are using.
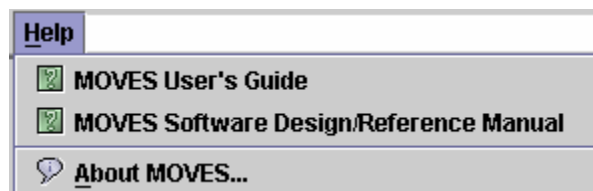
Click on "MOVES User's Guide" or "MOVES Software Design/Reference Manual" to open either of these documents in .pdf format in Adobe Acrobat Reader, or whatever Windows application is linked to the .pdf filetype extension on your system.

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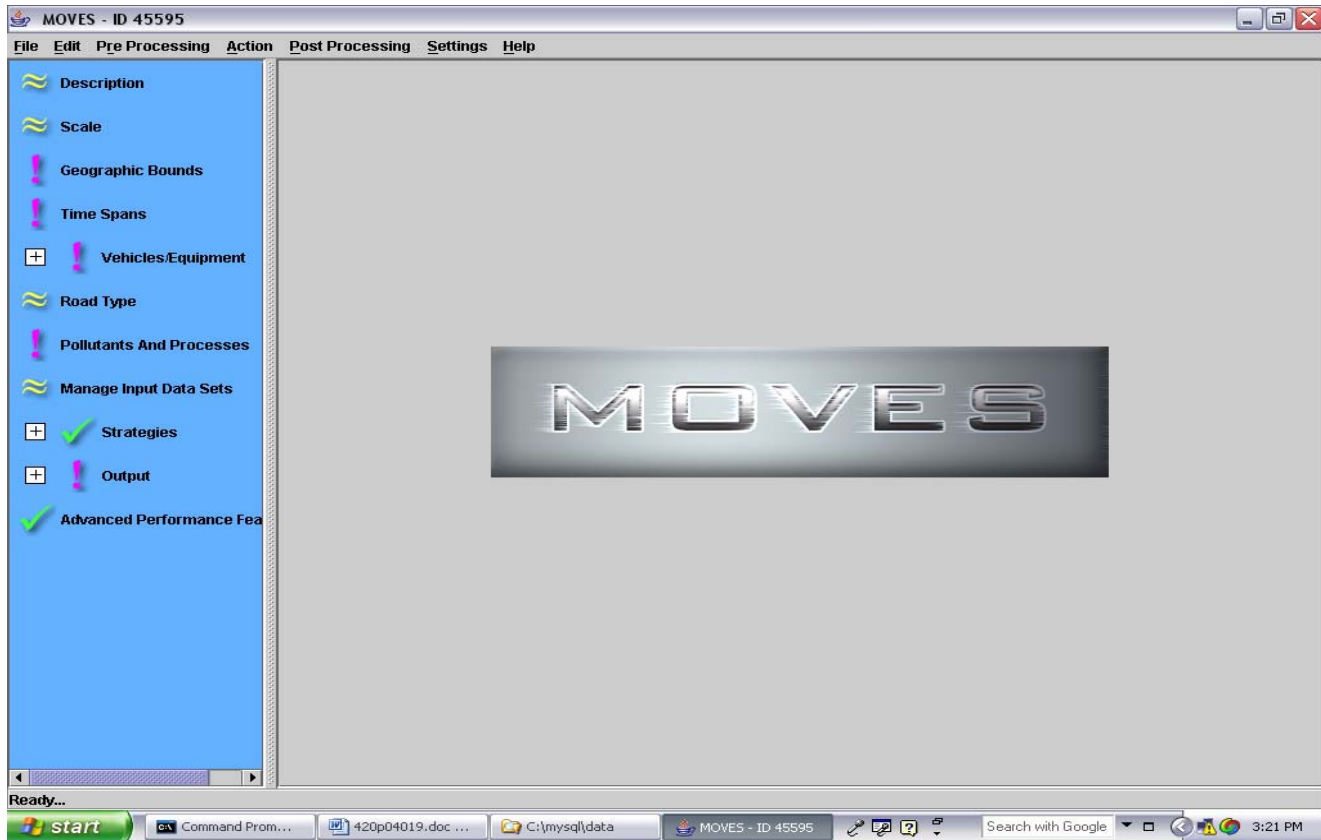
Selecting an item from the navigation list will place that item's detailed user interface into the detail panel on the right of the screen.

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:

!	**Needs additional user supplied data.**
✓	**Sufficiently filled in to run.**
≈	**Default data present, but otherwise sufficiently filled in to run.**
⊟	**TREE CLOSE/EXPAND**

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

Detail Panel

Detailed UIs 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 MOVES Worker program icon. (This step is now optional if a single-computer configuration is being used.)

ρ **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 program icon.

ρ **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.

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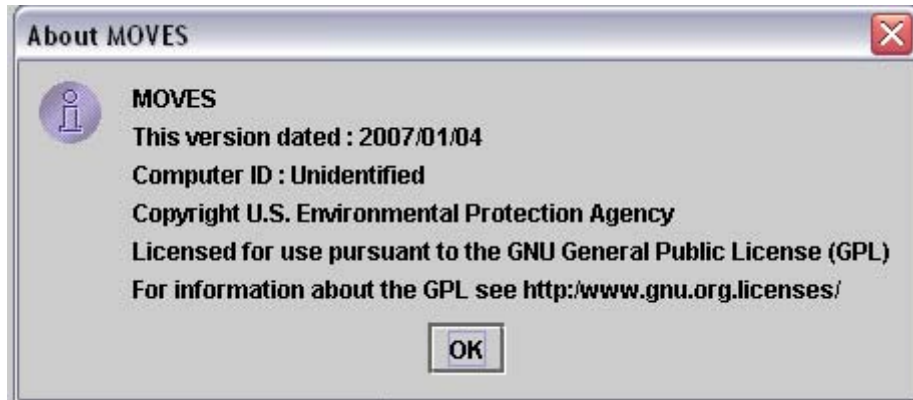
4. The MOVES RunSpec User Interface (UI) will open and input information may be entered. Entries may be made in any order, but a RunSpec cannot be executed until all the required information is provided.

Describing a 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. Its contents are reported for each run in the MOVESRun table.
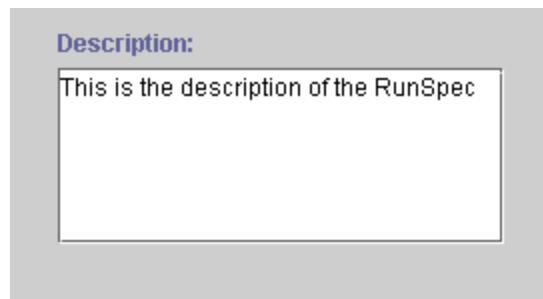
Up to 5000 characters of text may be entered to describe the RunSpec. In the above example "This is the description of the RunSpec" is the text. Text entered has no effect on the results from the MOVES program.

!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 chosen scale determines which geographic panel (e.g. Macroscale Geographic Bounds) will be presented to the user and may affect the operation of other panels as well. Currently, only the Macroscale and Mesoscale Lookup options are available. Macroscale is the default selection.

⚡CAUTION! Switching scales is a major change to the run specification. It affects the operation of other GUI detail panels and may cause their previous contents to be changed or lost.

You may wish to save your run specification before switching scales.

"Macroscale" is the default selection. The finest level of geographic detail available at this scale using the default input database is a kind of roadway (e.g., urban restriced access), or a single area location off of the highway network, within a county.

"Mesoscale Lookup" produces emission *rate* output, as opposed to emission *inventory* output. Emission rates are limited to those pollutants and processes which are attributable to actual roadways (e.g., running exhaust), as opposed to geographics zones (e.g., start exhaust).   The smallest level of geographic detail available at this scale using the default input database is the set of roadway segments in a county having the same average speed of vehicle travel.

The other two Scales, "Mesoscale" and "Microscale," 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 normally 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 of) all underlying data that is a function of geography (temperatures, road types, etc) 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 individual county 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 sub-section describes the three Macroscale options and the input needed for each choice.

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 particular States or Territories within the United States. If the "*State*" button is selected a new bounds panel will show in the Detail Panel, as shown below.

This panel has 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.

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

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.

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.

Zone and Link

A fourth geographic setting, "Zone and Link", is active for "Mesoscale Lookup" and is the only valid selection at this scale.

The geographic selection panel for this option operates as it does at the "County" setting. The "Zone and Link" setting represents the fact that when the user selects a County at this scale, all zones and links in the County are included.

Selecting Time Spans

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

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 of) all underlying MOVES internal data that are segregated by hour (i.e., temperatures, VMT distributions, etc) prior to execution of the run.   For example, if the user chooses "*Month*" the model will average the temperatures of all selected Days and Hours into an average "*Month*" set (it 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, kinds of days, or months. If such a de-selection is done, the user will end up with results that are based on aggregations that do not include de-selected hours, days, or months. For example, if the user selects "*Months*" in the Time Aggregation Level panel and de-selects "Weekends" the final emission result will represent a monthly aggregation that includes "Weekdays" only.

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 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.

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 kinds of days of the week to model to select the appropriate boxes in the DAYS section. At least one

kind of Day box must be selected. Click *"Select All"* button to select all of the days. Similarly, the *"Clear All"* button removes all of the previously chosen days of the week.

Hours

The default time resolution for MOVES at both Macroscale and Mesoscale Lookup 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 selected hours of the day.

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

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

NOTE The hours selected are run for all chosen days, so it is not possible, for example, to model only from 11:00 PM on Friday to 01:59 AM on Saturday. It is also not possible to run the model for a group of months that span a year end. That is, the user cannot run the model for just December of 2003 through January 2004. However, the model can be run multiple times with different RunSpec time spans to

accomplish these tasks, or it can be run for multiple days or years and then manually post-processed to eliminate the un-needed times.

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 selections for the vehicles/equipment.

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** The On Road Vehicle Equipment Requirements Panel contains a small subpanel at the bottom that reminds the user that they are required to select a Fuel and Source Use type.   This panel disappears after the user makes a valid Fuel and Source Use type selection. If a selected combination is not in the database, the subpanel will display this information. No results are reported for such vehicles.

Selecting RoadType

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.

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

1. Click and highlight the desired ROAD TYPES from the scroll down list.

2. Click "*Add*" button to 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 ROAD TYPES in the scroll down list.

NOTE The scroll down list of ROAD TYPES is the complete list of roadway types present in the underlying MOVES database. The default database has for Road Types which represent urban and rural driving on roads with restricted and

unrestricted vehicle access. Restricted vehicle access is usually used to model freeways and interstates

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

⚡CAUTION! Entries to the ROAD TYPES list are <u>not</u> 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.

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

	Running Exhaust	Start Exhaust	Crankcase	Evap Refueling Losses	Brakewear	Tirewear	Evap Permeation
Total Gaseous Hydrocarbons	✔	☐					☐
Carbon Monoxide (CO)	☐	☐					
Oxides of Nitrogen	☐	☐					
Primary PM10 - Organic Carbon							
Primary PM10 - Elemental Carbon							
Primary PM10 - Sulfate Particulate	☐	☐					
Primary PM10 - Brakewear Particulate							
Primary PM10 - Tirewear Particulate							
Primary PM2.5 - Organic Carbon	☐	☐					
Primary PM2.5 - Elemental Carbon	☐	☐					
Primary PM2.5 - Sulfate Particulate	☐	☐					
Primary PM2.5 - Brakewear Particulate					☐		
Primary PM2.5 - Tirewear Particulate						☐	
Total Energy Consumption	☐	☐					
Petroleum Energy Consumption	☐	☐					
Fossil Fuel Energy Consumption	☐	☐					
Methane (CH4)	☐	☐					
Nitrous Oxide (N2O)	☐	☐					
Atmospheric CO2	☐	☐					
CO2 Equivalent	☐	☐					

☑ **Distance Traveled**

Pollutant/Process Requirements

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** It may be necessary to scroll the pollutant-process detail panel to see all of the process columns.

⚲ **NOTE** 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 D ISTANCE T RAVELED check-box to have the MOVES model compute and report the distance traveled by
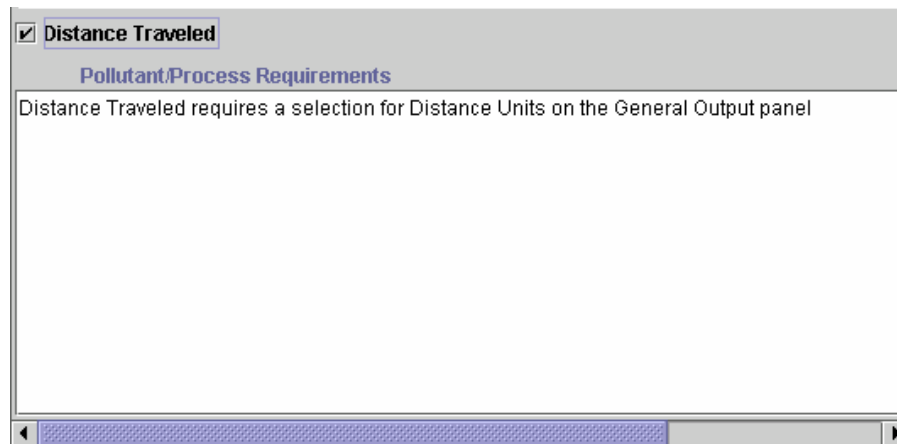
vehicles in the selected geographic, roadway, or time strata.

 N OTE The D ISTANCE T RAVELED box will be 'grayed out' if no Running Exhaust Process is chosen in P OLLUTANTS AND P ROCESS selection panel. It also requires a selection for D ISTANCE U NITS on the G ENERAL O UTPUT panel.

 N OTE The P OLLUTANT /P ROCESS R EQUIREMENTS 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.

 N OTE Columns are included in the panel for Crankcase and Evaporative Refueling Loss Emissions, because these emission processes are planned to be included in

subsequent versions of MOVES-HVI. The demonstration version, however, does not estimate these emissions and so there are no pollutant choices available in these columns in this version.

The pollutants and processes included in MOVES-HVI are described briefly here:

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.

e.   Evaporative Fuel Permeation — the migration of hydrocarbons through the various elastomers in a vehicle fuel system.

f. Evaporative Fuel Vapor Venting — the expulsion into the atmosphere of fuel vapor generated from evaporation of fuel in the tank. Also includes evaporation into the atmosphere of fuel which has "seeped" to the surface of vehicle parts.

g. Evaporative Fuel Leaking — the "gross" leaking of fuel, in liquid form, from the vehicle. This is assumed to subsequently evaporate, outside the vehicle, into the atmosphere.

h. Brakewear — the formation of particles of brake components which are formed during operation of vehicle brakes.

i. Tirewear — the formation of tire material particles during vehicle operation.

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 (N_2O) 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.

f. Atmospheric CO_2 – This pollutant is in units of mass. The designation "atmospheric" refers to the fact that this includes other gaseous carbon components such as CO and THC which are assumed to eventually react in the atmosphere to produce CO_2.

g. CO_2 Equivalent – This "pollutant" is in units of mass and represents the combined emissions of CO_2, methane and N_2O weighted by their "global warming potential" values. CO_2 has a global warming potential value of 1.0.

h. Total Gaseous Hydrocarbons (THC) — This pollutant is in units of mass.

i. Carbon Monoxide (CO) — This pollutant is in units of mass.

j. Oxides of Nitrogen (NOx) — This pollutant is in units of mass.

k. Primary PM2.5 – Organic Carbon. This pollutant consists of organic carbon particulate matter having particle size less than 2.5 microns. This pollutant is in units of mass. The designation of "primary" refers to the fact that the quantity reported is what is emitted from the vehicle, and does not take into account subsequent chemical reactions in the atmosphere.

l. Primary PM2.5 – Elemental Carbon. This pollutant consists of elemental carbon particulate matter having particle size less than 2.5 microns. This pollutant is in units of mass. The designation of "primary" refers to the fact that the quantity reported is what is emitted from the vehicle, and does not take into account subsequent chemical reactions in the atmosphere.

m. Primary PM2.5 – Sulfate. This pollutant consists of sulfate particulate matter having particle size less than 2.5 microns. This pollutant is in units of mass. The designation of "primary" refers to the fact that the quantity reported is what is emitted from the vehicle, and does not take into account subsequent chemical reactions in the atmosphere.

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.

!Tɪᴘ   A selection for Dɪsᴛᴀɴᴄᴇ Uɴɪᴛs on the Gᴇɴᴇʀᴀʟ Oᴜᴛᴘᴜᴛ panel is also required for this computation.

A message appears in the Pᴏʟʟᴜᴛᴀɴᴛ/Pʀᴏᴄᴇss Rᴇᴏᴜɪʀᴇᴍᴇɴᴛs 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 a table containing only 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.

ϙ **NOTE** These databases <u>must</u> adhere to the MOVES schema and use the database management system (DBMS) used by MOVES. In practice this means that individual tables in the user-supplied database must have identical names as the 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 MOVESHVI Software Design Reference Manual.

⚷ **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 two alternate databases called "UserInputDB1 and "UserInputDB2" being added as MOVES inputs. The unspecified Server name implies that the database resides in the user's local computer in the MySQL Data folder.

1. Type a new database name and click the "*Create Database*" button to create a new database, with empty tables, on an existing local or remote server. Or select an existing database from the dropdown list.
2. You can use database tools, outside of MOVES to populate new database tables with alternate values. Within MOVES the I/M Table Coverage Editor and the FERC also create and populate User Input databases.
3. Click "*Add*" button to add the new or existing database to the input databases, shown in the SELECTIONS window. MOVES will first validate that the combination of server and database

are unique within the selections. The same server-database combination cannot be added more than once.

> **!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.

4. 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 previous ones in cases where the internal tables and records of each database are the same.

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

The mechanism to specify input databases for the true Mesoscale and the Microscale options on the MOVES 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. (The Mesoscale Lookup scale option does not require this mechanism.)

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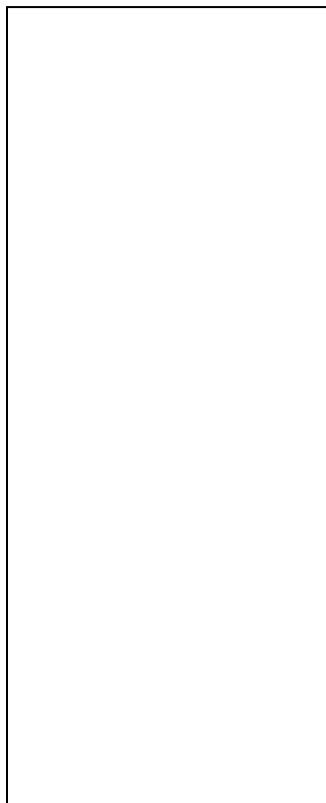
Click buttons on the right side of the panel to manipulate the strategies needed for each RunSpec, as detailed below. The strategy name will appear in the window under LOADED OBJECTS.

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 AVFT file must also be selected from a previous "*Export*" of a Strategy.

ꝙ **NOTE** These exported stategy files available for import, are **not** the same as those saved within RunSpecs. They store only a single, constant set of settings.

⚡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:
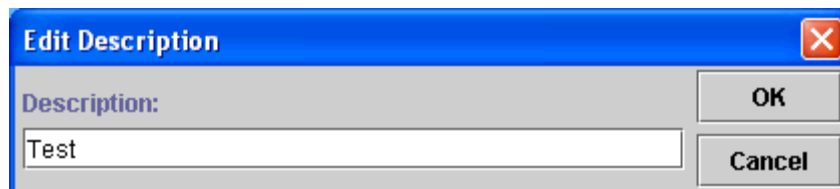
Descriptions can never be completely blank (they wouldn't show on screen then), so entering a blank description results in the phrase "*(default)*" being displayed instead. This default description does not necessarily mean that the strategy contains only its default data.

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. 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 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.

Loaded objects:							

Source Type:

	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 ▼	Normalize	Add Model Year

	Gasoline Conventional Internal Combustion	Diesel Fuel Conventional Internal Combustion	Advanced Gasoline <<	Gasoline Advanced Internal Combustion	Gasoline Conventional Internal Combustion - Hybrid - Mild	Gasoline Conventional Internal Combustion - Hybrid - Full	Gasoline Advanced Internal Combustion Hybrid Mild	Gasoline Advanced Internal Combustion Hybrid Full	Advanced Diesel >>
2000	99.70%	0.08%	0.18%	0.03%	0.08%	0.08%	0.00%	0.00%	0.0
2001	99.70%	0.08%	0.18%	0.03%	0.08%	0.08%	0.00%	0.00%	0.0
2002	99.51%	0.07%	0.38%	0.14%	0.12%	0.12%	0.00%	0.00%	0
2003	99.10%	0.10%	0.77%	0.27%	0.25%	0.25%	0.00%	0.00%	0
	96.37%	0.18%	3.41%	0.54%	1.43%				

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.
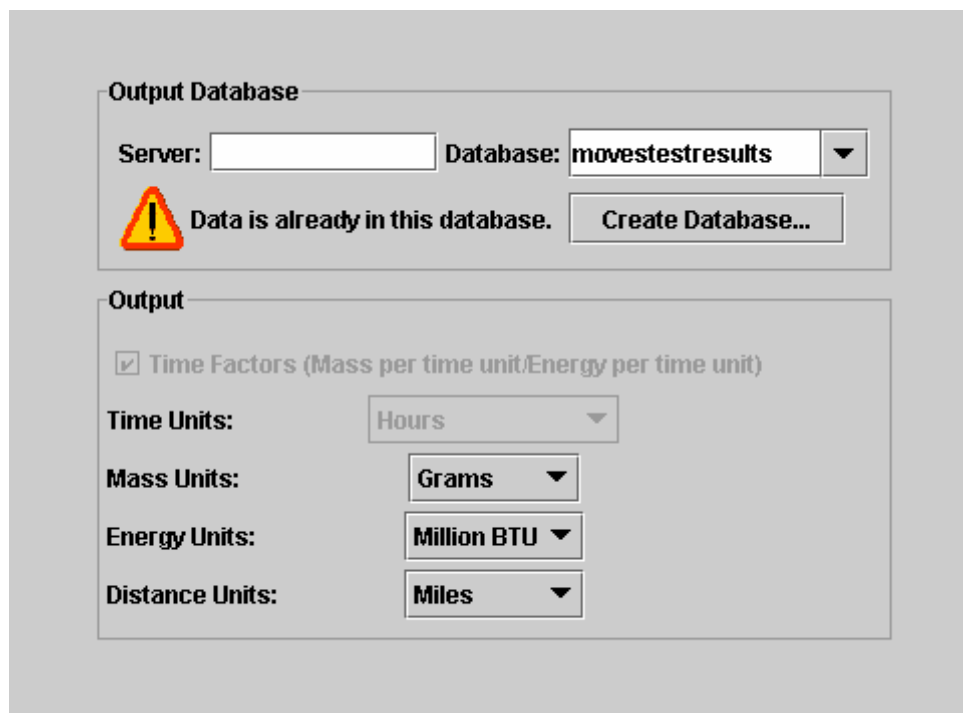
Specifying Output Dimensions

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 aspects 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.

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 may click the Create Database... button. All new output databases begin with a MOVES Run identification number of one.

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

Choosing Units for Mass, Energy, and Distance
The second sub-panel is the OUTPUT sub-panel. It contains user choices for the mass, energy, and distance units in which the results will be reported.   The time units are set to correspond to the time reporting level (see next section) and is

displayed, but is not user-selectable on this panel.   The possible choices for mass are kilograms, grams, pounds, or US tons (short tons). The possible energy units are Joules, KiloJoules, or Million BTUs (British Thermal Units). The possible distance units are miles or kilometers. Distance units may be selected only if "distance" output has been requested on the pollutants and processes panel. The user may choose only one unit for each of the four types. The units are reported in the output database in the MySQL output table "movesrun."

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 <u>rate</u> such as kilograms per hour. Because the time units is set equal to the time reporting level this quantity amounts to a total inventory quantity mass of emissions or total energy over the output time period.

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

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

Always Box

The Aʟᴡᴀʏs box at the upper left is a reminder that the output data will always contain dimensions for time, location, and pollutant. The Aʟᴡᴀʏs box also contains two buttons that

allow the user to specify the level of output aggregation by time and location level.

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

The "Portion of Week" selection requires some explanation: As we saw on the "Timespans" panel the "Day" selections there do not necessarily select a particular day of the week such as Monday, Tuesday, etc where one day follows another, but just an unordered category representing a *kind of day of the week.* Default databases produced by EPA are likely to include only a single kind of day, or may have two kinds of days, weekdays and weekend days. In the case where the MOVES Input database just has a single kind of day, then reporting by "Portion of Week" would report for the entire 7-day week.   In the case where the MOVES Input database has two kinds of days, representing week days and weekend days, then reporting by "Portion of Week" would report totals for the 5 weekday period and for the 2 weekend day period, assuming that both kinds of days were included in the run specification.

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 MOVES-HVI includes only On Road Vehicles. (The ON ROAD section is always available and the OFF ROAD section is always grayed out in MOVES-HVI.)

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.

SourceUseTypes are native to MOVES, so this is the

preferred option. MOVES can report by Source Classification Code (SCC), but this introduces an additional approximation step in the calculations to convert output by SourceUseType to SCC.

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

Estimate Uncertainty

The "Estimate Uncertainty" box should be left unchecked by most users. When checked the remainder of the uncertainty panel becomes active for the user to specify a number of iterations, whether randomly sampled input used for each iteration should be reported, and whether the output produced from each iteration should be reported. This mechanism may be used to investigate the uncertainty of MOVES results due to uncertainty in some of its inputs.   Users should consult section 10.38 of the MOVES-HVI Software Design and Reference Manual before attempting to use this feature.   The default database provided with MOVES-HVI does not contain realistic input data for this feature.

↗**CAUTION!** Model execution time and the volume of output produced (if the output of each iteration is saved) are generally increased in proportion to the number of iterations

specified. If the randomly sampled inputs for each iteration are saved as well, even more voluminous output is produced.

Advanced Performance Features

This Navigation Panel item invokes features which may be used to diagnose problems with the model software or to improve the execution run time performance of long model runs by saving and avoiding unnecessary recalculations (intermediate results). Use of these features requires knowledge of the detailed software components of the MOVES program, what inputs they need, and what outputs they produce. Users interested in this feature should review the MOVES-HVI Software Design and Reference Manual.

Masterloopable Components

Component	Don't Execute	Save Data
Total Activity Generator (TAG)	☐	☐
Operating Mode Distribution Generator (running OMDG)	☐	☐
Start Operating Mode Distribution Generator	☐	☐
Source Bin Distribution Generator (SBDG)	☐	☐
Meteorology Generator	☐	☐
Tank Temperature Generator	☐	☐
Tank Fuel Generator	☐	☐
Mesoscale Lookup Total Activity Generator	☐	☐
Mesoscale Lookup Operating Mode Distribution Generator	☐	☐
Emission Calculators	☐	☐

Destination User Dataset

☐ Copy Saved Generator Data

Server:

Database: ▼ Create Database

3. Executing Example MOVES2004 Run Specification

The example MOVES2004 Run Specification models the entire U.S for one year (1999) at the most aggregate level. It estimates total energy consumption and emissions of methane and nitrous oxide for the running, start, 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.)

Step-by-step Example Run

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 MOVESExample.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 6 minutes to execute on a single computer configuration, using a PC with multiple processors, a gigabyte RAM and a processor speed of 2.1 GHz.

6. The output will appear in a MySQL relational database. The name of the database will be the same one specified Step 4.

MOVES Output Structure

The database will typically contain five 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.

MovesWorkersUsed – This table is not of concern to most users.   It contains information as to which MOVES Worker Program copies processed portions of the run. This is only of interest if the run is executed on a multi-computer configuration.

4. Customizing MOVES

Although a full range of default data is available in MOVES, the model is 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.

Several components of the model are intended to facilitate customization.   These include the IM Coverage Table Editor described in Appendix A, the Future Emission Rate Creator described starting on page 11, and the ability to select Strategies described beginning on page 55.   The ability to provide User Input Databases, described on page 52, while requiring the user to prepare MySQL database(s), is very powerful because it allows all model inputs to be changed.

Local Customization

One approach to customizing MOVES for a specific area is to redefine the MOVES domain from the default "entire US plus territories" to a smaller domain such as a single state. At a minimum, this approach requires changing the base year VMT values to the totals needed for the new domain and changing the geographic (zone-level) allocation factors to properly allocate the new VMT among counties or other sub-domains. If not changed, the other default fleet and activity measures in the database,

such as allocations by time, roadway type, age and sourcetype, will be applied to the new domain.

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 and accurate the data for an area, the better the output will be for that area.

"What-if" Analysis Customization

MOVES-HVI provides a very broad range of "what-if" analysis capability for looking at different future scenarios. "What-if" capability extends to many dimensions, e.g. fuel and vehicle technology penetration, energy and emission performance of these technologies, 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 makes the necessary

changes to the underlying MOVES database tables that feed 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).

Activity Patterns

MOVES provides unprecedented flexibility for modeling vehicle activity patterns. User who want to examine the emission impact of hypothetical changes in VMT growth or allocation, or in driving patersn may directly alter these values in the underlying MySQL database. This should be done through the "Mange User Input Data Sets" screen of the MOVES graphical user interace. Users will want to consult the SDRM and the "Highway Vehicle Population and Activity" reports for information on specific fleet and activity parameters.   For "What if" analysis users might want to look at <u>non</u> real-world activity – hypothetical VMT increases or decreases, hypothetical changes

in average driving speed, etc. The user can alter such inputs directly in the underlying MySQL database.

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

Appendix A. Displaying and Editing Vehicle Inspection and Maintenance (I/M) Program Coverage Information

The I/M Table Coverage Editor makes it easy to display and modify the vehicle inspection maintenance (I/M) program coverage information in the MOVES database(s) that are to be used by the current run specification. For the editor to operate properly the run specification must specify the calendar years, locations (states or counties), source use types, and pollutant-processes of interest.

This editor is accessed by selecting "Display/Edit IM Program Coverage Records" from the "Pre-Processing" menu:
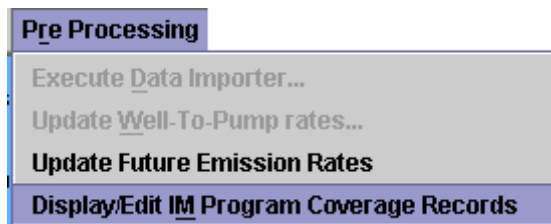
When selected a popup menu displays the I/M program information pertinent to the run specification:

A printed report of this information may be produced by clicking on the "Print Report" button.
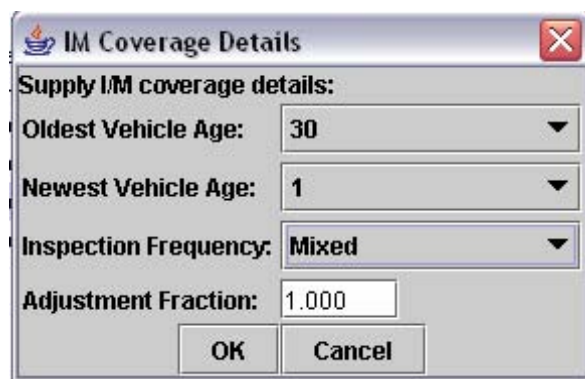
The user may select one or more records in this display. (In the example shown above one record, pertaining to the coverage of light duty trucks, has been selected.) When one or more records are selected the "Edit Selected Records" button becomes active and may be used to change the data values in the selected records. If this is clicked a popup appears asking the user to "Choose a User Input Database":
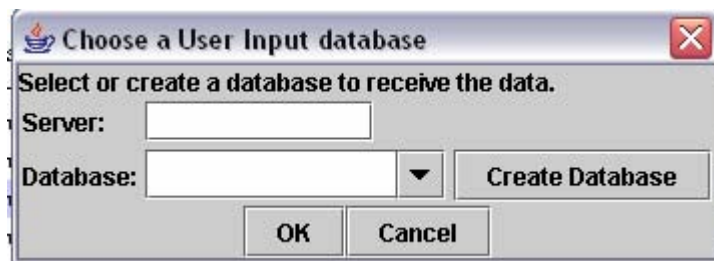
The user should normally leave the server name field blank. An existing database, which should contain an IMCoverage Table, can be specified in the database field, or a new database name can be entered and created by clicking on the "Create Database" button. (If this is done a popup appears confirming that the database was created; click on "OK" closes this popup.)

Clicking on "Cancel" aborts the editing operation; clicking on "OK" would be the normal next step and causes the program to display another popup asking the user to supply data values for the record(s) to be changed. If all selected records have the

same data value, (as in the example shown since only one record is selected), this is displayed as the default data value choice. If the selected records have different values of a data field, the input field is left blank and the user must supply a new value.

Dropdown lists can be used to select a new value for the oldest and youngest vehicles covered by the program, and the inspection frequency which may be Annual, Bienniel, Continuous, or Mixed (indicating that different model year vehicles are inspected at different frequencies). A new value may also be entered for the "Adjustment Fraction".

In this example we change the Inspection Frequency for the selected record to be "annual" and the Adjustment Fraction to be 0.9. (An adjustment fraction value of 1.0 means the program is fully effective. Smaller values reduce the benefit proportionately; a zero value would mean the program has no effect.)

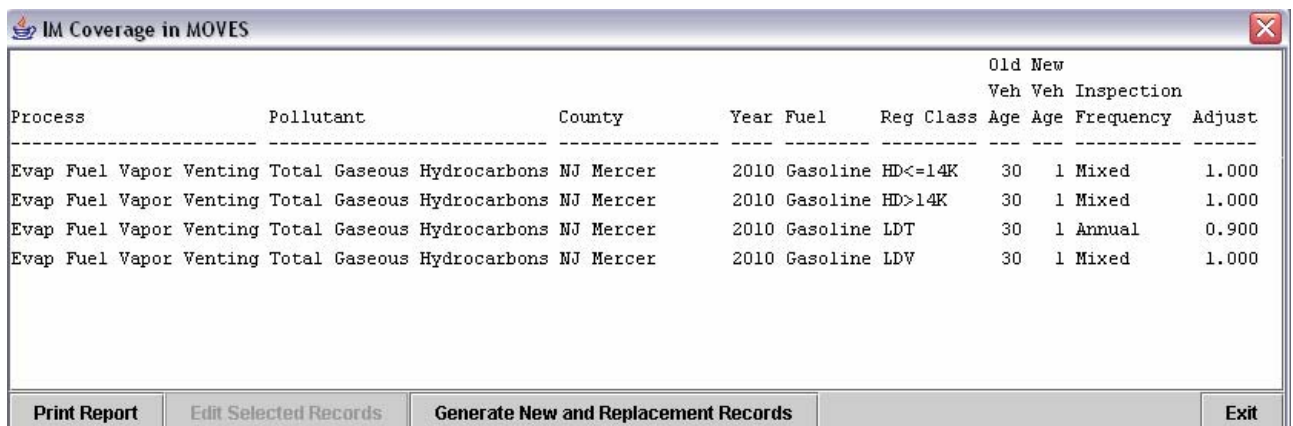**IM Coverage Details**

Supply I/M coverage details:

| Field	Value
Oldest Vehicle Age:	30
Newest Vehicle Age:	1
Inspection Frequency:	Annual
Adjustment Fraction:	0.9

OK   Cancel

Clicking on "Cancel" aborts the editing operation. Clicking on "OK" makes the specified change and displays the record information showing the changes.

IM Coverage in MOVES

Process	Pollutant	County	Year	Fuel	Reg Class	Old Veh Age	New Veh Age	Inspection Frequency	Adjust
Evap Fuel Vapor Venting	Total Gaseous Hydrocarbons	NJ Mercer	2010	Gasoline	HD<=14K	30	1	Mixed	1.000
Evap Fuel Vapor Venting	Total Gaseous Hydrocarbons	NJ Mercer	2010	Gasoline	HD>14K	30	1	Mixed	1.000
Evap Fuel Vapor Venting	Total Gaseous Hydrocarbons	NJ Mercer	2010	Gasoline	LDT	30	1	Annual	0.900
Evap Fuel Vapor Venting	Total Gaseous Hydrocarbons	NJ Mercer	2010	Gasoline	LDV	30	1	Mixed	1.000

Print Report   Edit Selected Records   Generate New and Replacement Records   Exit

Since the "Edit Selected Records" process cannot be used to create new records, the "Generate New and Replacement Records" feature is also provided. When this is selected a popup window appears asking the user to "Choose A User Input Database" just as for the "Edit Selected Records" option. Once a database has been identified a new popup window appears asking the user to specify the set of records they would like to create:
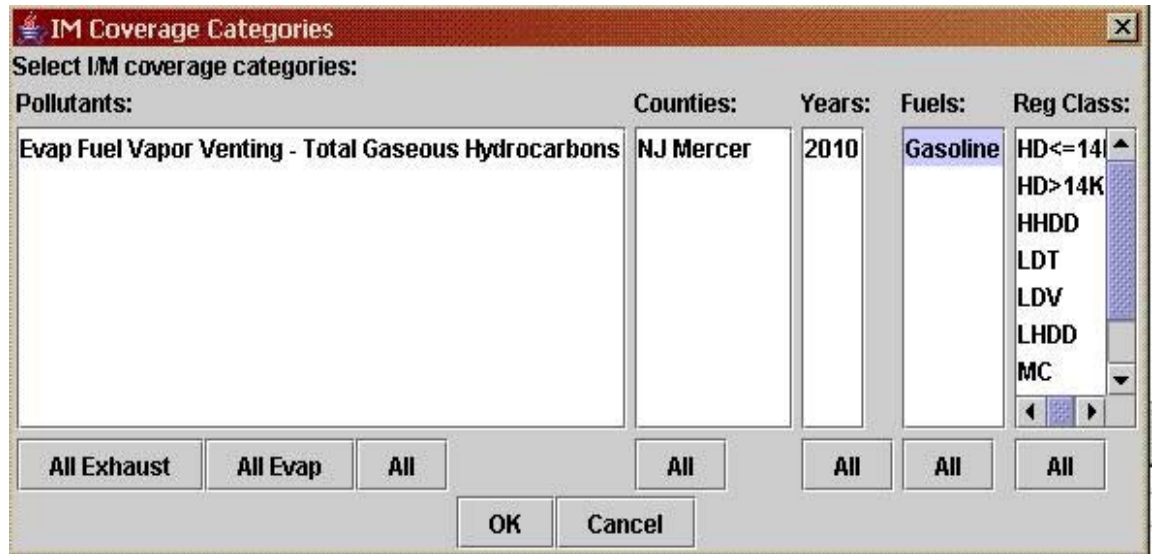
IM Coverage records are specified by pollutant-process, location (County), calendar year, fuel type (currently just gasoline-fueled vehicles are covered by IM programs), and regulatory class.   The choices offered in each area are restricted to those included in the current run specification and which are relevant to IM. In this example the IM program coverage information is already as complete as it can be for the underlying run specification (which has limited the run to Evap Vapor Venting emissions in Mercer County, NJ, for the year 2010). So we would "cancel" to abort this editing operation.

Normally the user selects one or more values in each category and clicks "OK".   This causes the same popup window prompting for data values for the new records to appear as in the "Edit Selected Values" option.   Once new data values are provided, the display is updated.

Either of these editing processes may be repeated any number of times. When the user eventually clicks on "exit" the User Input Database is written and added, if necessary, to the run specification.

Appendix B. Linking MySQL tables from MS ACCESS or MS Excel (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. This appendix also explains how to use the Open Database Connectivity interface protocols and tools.

Adding a MySQL database as an ODBC User Data Source

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

2. Create an ODBC data source 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) :
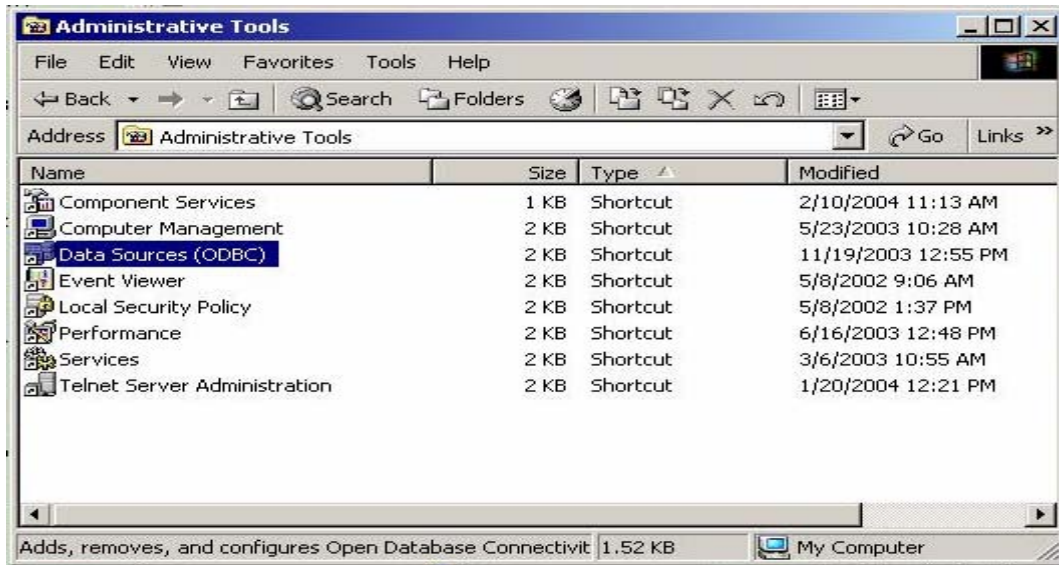
(iii.)   In the ODBC Data Source Administrator dialog choose to "Add"
          a "User Data Source"
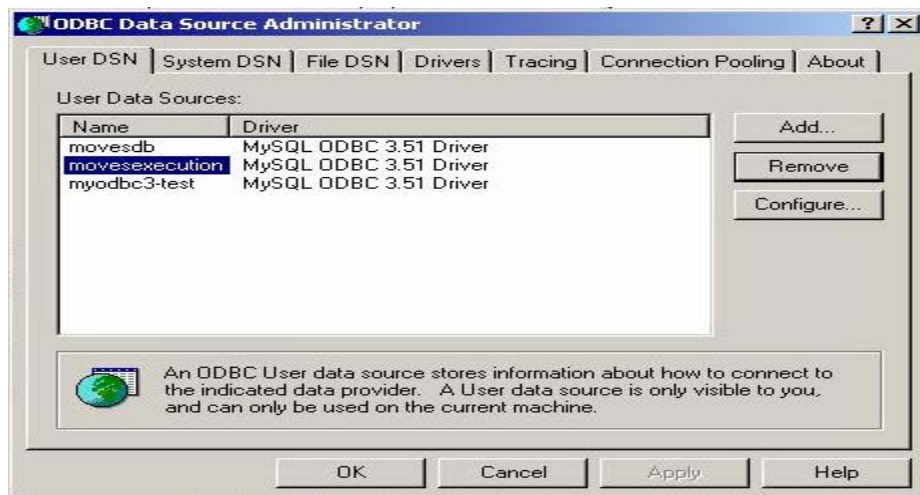
(iv.) The "New Data Source" should use the MySQL ODBC Driver that was installed earlier. So, scroll through the list until the MySQL ODBC driver is found, highlight it, and click the "Finish" button.

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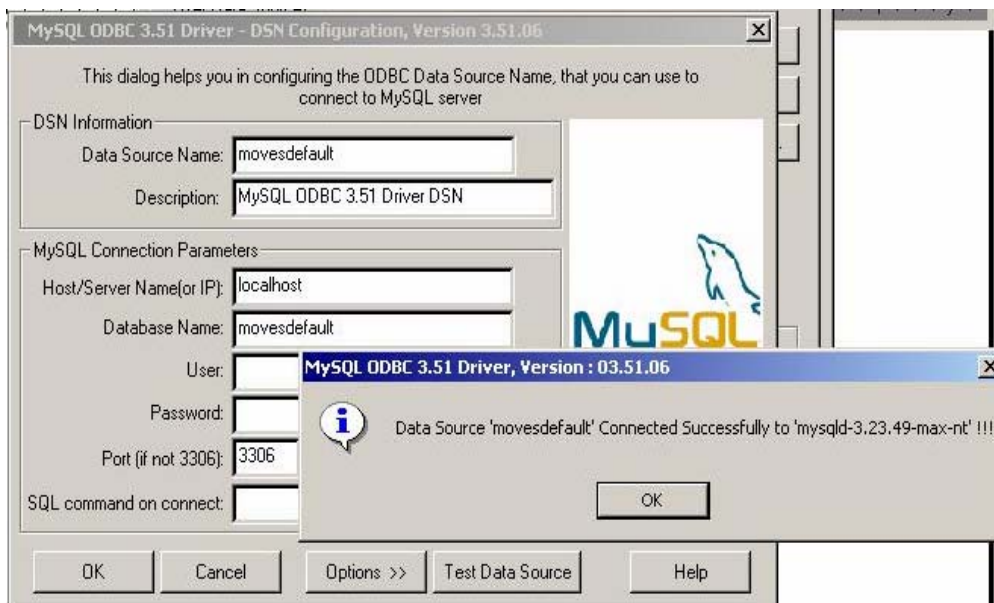
(vii.)   The added data source will show up in the list of data sources in the ODBC Data Source Administrator dialog. And this will finish the process of creating a new ODBC data source associated with a particular MySQL database.
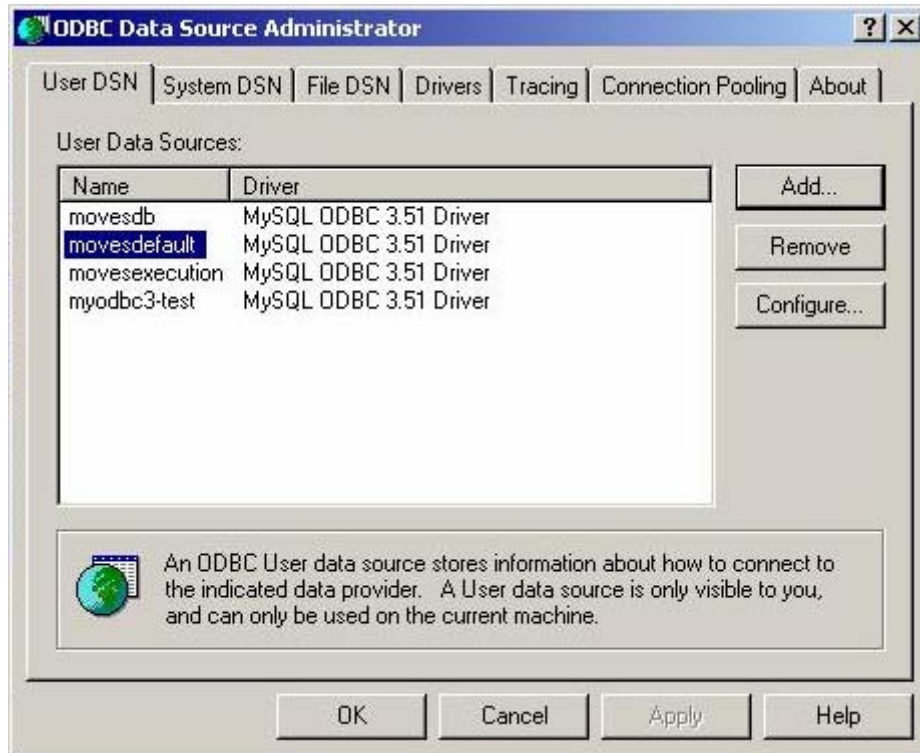
Using Microsoft Access with MySQL Tables

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

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

℺ **NOTE** Using "Import" instead of "Link Tables" in Microsoft Access 97: There is a known problem with Microsoft Access 97 when linking to 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 are several possible work-arounds to overcome this problem, however using the "Import" feature is the simplest one to use.
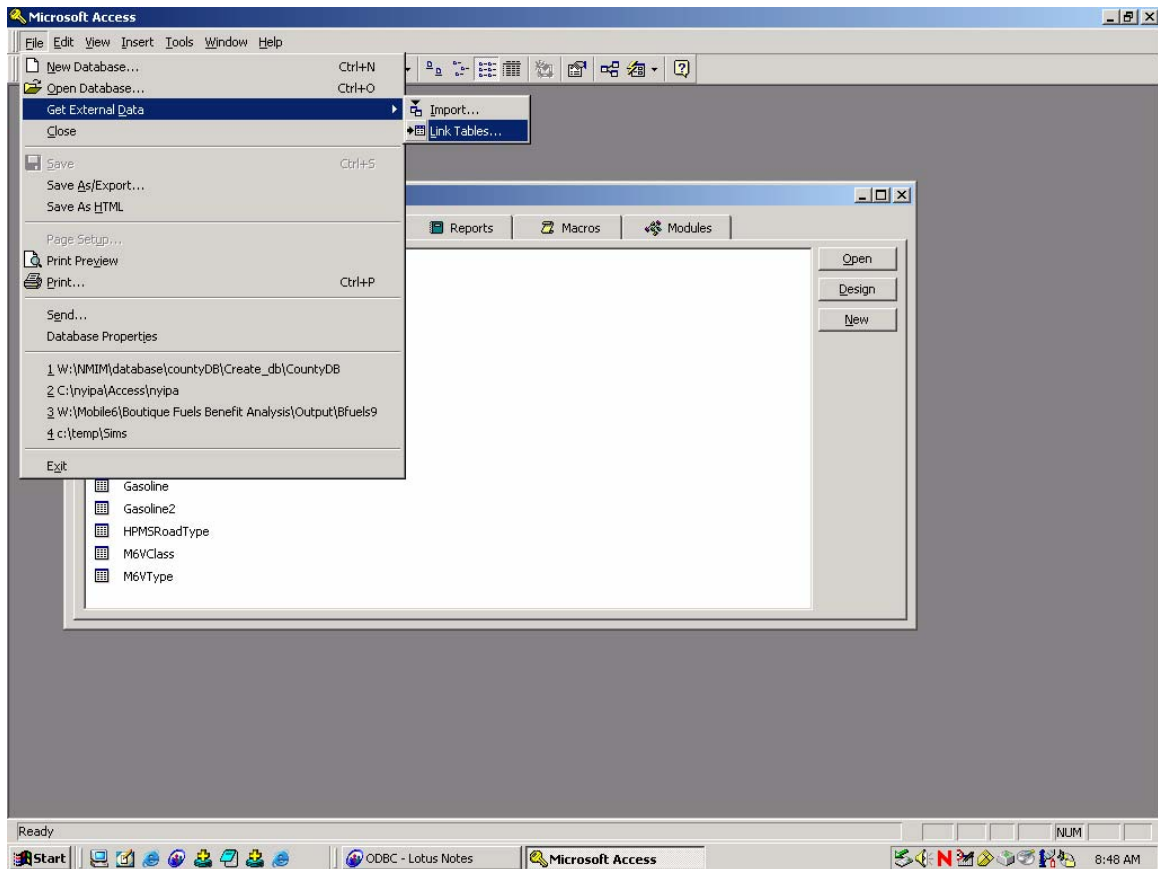
3. On the pop-up windows like the one below.
Choose "ODBC Databases()" from the drop-down list which is located next to the label "Files of type" (you may need to scroll down a bit from this drop-down list box to find "ODBC Databases()").

Link dialog box

| Look in: | My Documents |

Corel User Files
My Pictures
Security

Link
Cancel
Advanced...

Find files that match these search criteria:

File name: [   ]   Text or property: [   ]   Find Now

Files of type: Microsoft Access (*.mdb;*.mdw;*.mda)   Last modified: any time   New Search

dBASE IV (*.dbf)
Microsoft Excel (*.xls)
Microsoft FoxPro (*.dbf)
HTML Documents (*.html;*.htm)
Text Files (*.txt;*.csv;*.tab;*.asc)
ODBC Databases ()

Not all file typ... e types, run the Setup program, click Add/Remove, Data Access,
Change Optio... the Office 97 ValuPack.

0 file(s) fou...

4. Another pop-up window similar to the one below will appear. Click on the tab "Machine Data Source"; select a database of interest, in this example we use "CountyDB", from the "Data Source Name" column. (if you can not find "CountyDB" under the "Data Source Name" column, try to find it under tab "File Data Source").
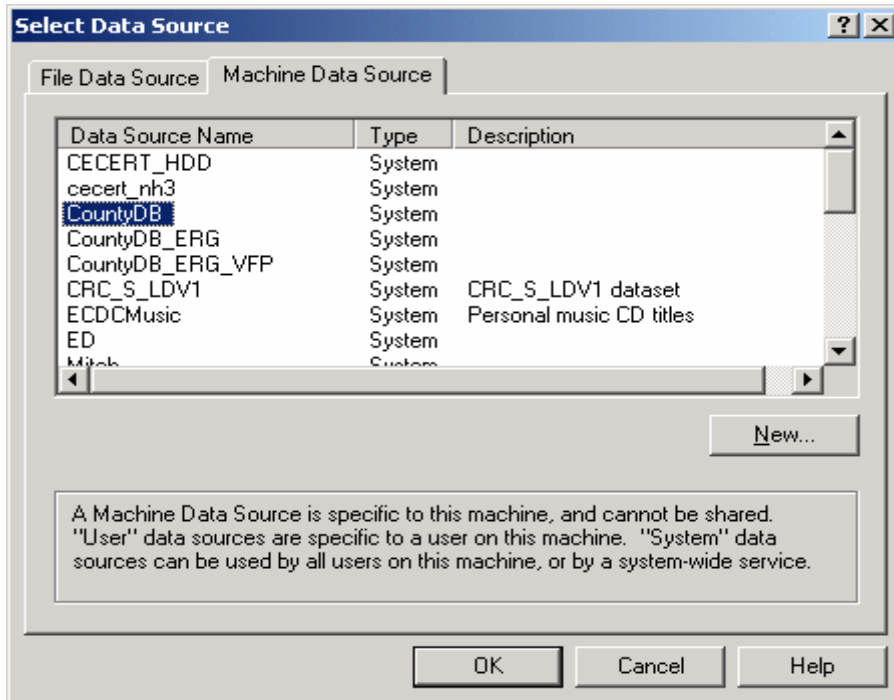
Click on the "OK" button to invoke the ODBC window,
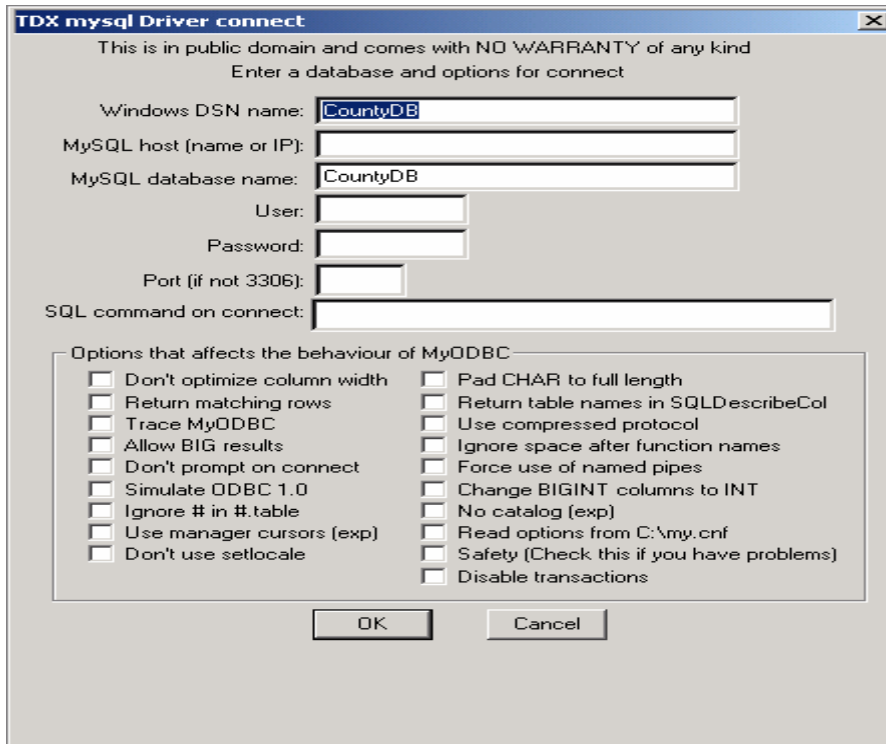
5. Do nothing, just click on "OK" button.

6. On the "Link Tables" widow 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 the tab " Tables".
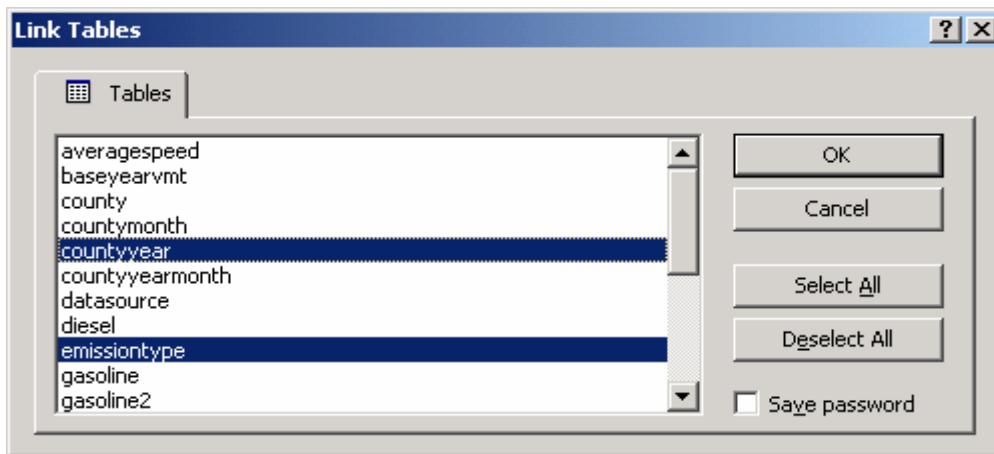
You can now work with these tables in MS Access. You can view and query the tables normally.

✔ **CAUTION!** You may edit these tables as well, but because you have "imported" them you are working with separate tables in MS ACCESS. Any changes you make will not affect the original MySQL tables.

7. To remove 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.
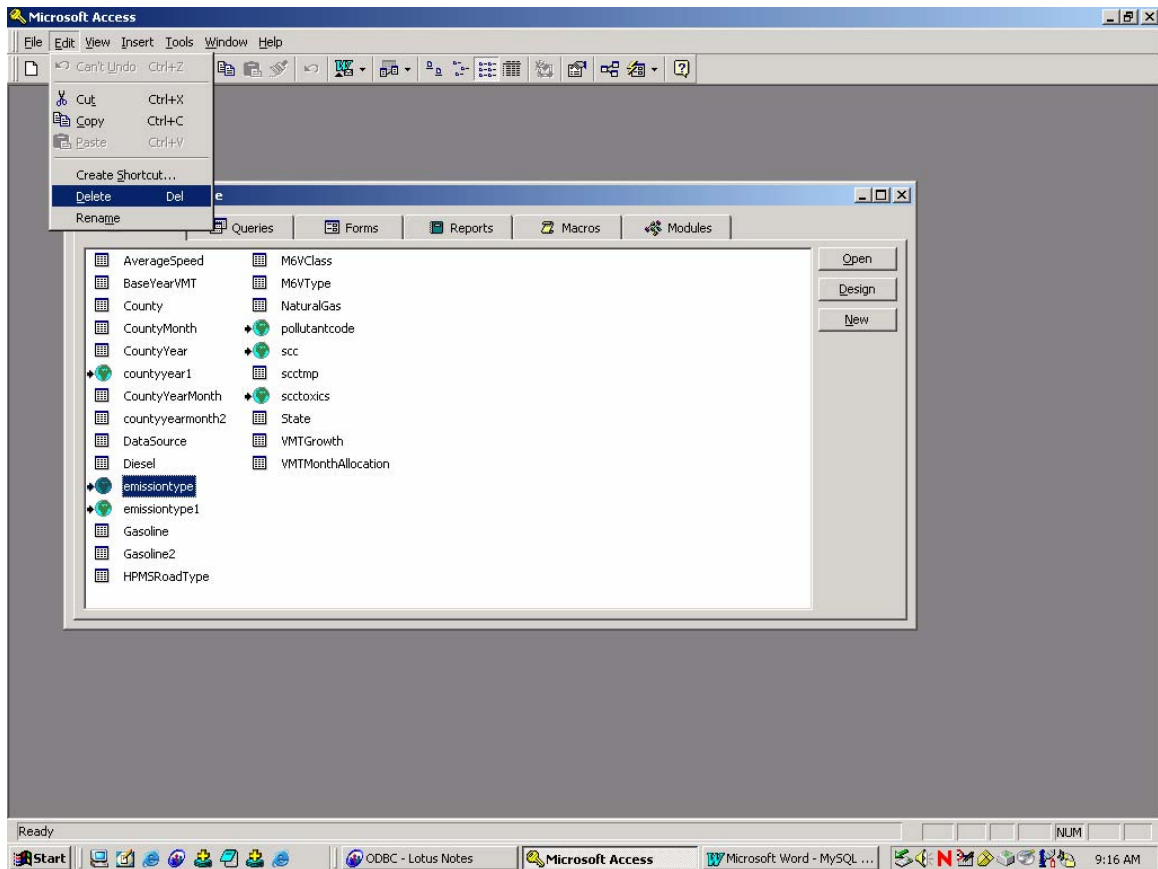
Using Microsoft Excel 97 with MySQL Tables

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

2. Open an Excel file; click on Data from the top menu bar; 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 "Choose Data Source" window.

3. From the Choose Data Source window, select the desired DSN from the list, e.g., "MOVESDB", then click on OK button.

Choose Data Source

Databases | Queries

Megan_Test*
MOVESDB*
MOVESDB2004*
MOVESDefault*
MQIS (not sharable)
MQIS*
MS SQL Server6-Approach (not sharable)
MS Sybase SQL Server-Approach (not sharable)
MSOD_Pub*
NMIMOUTPUT*

OK
Cancel
Browse...
Options...

☑ Use the Query Wizard to create/edit queries
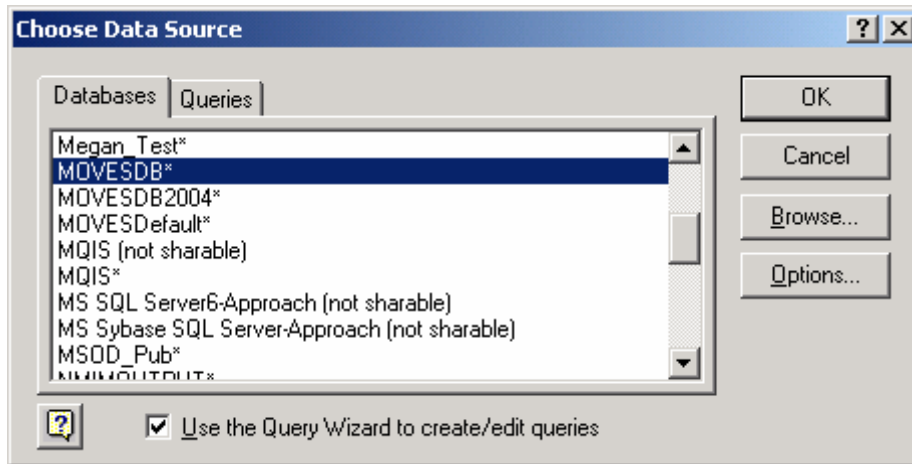
4. Click on the OK button in the "mysql Driver connect" window

TDX mysql Driver connect

This is in public domain and comes with NO WARRANTY of any kind
Enter a database and options for connect

Windows DSN name: MOVESDB

MySQL host (name or IP):

MySQL database name: MOVESDB

User:

Password:

Port (if not 3306):

SQL command on connect:

Options that affects the behaviour of MyODBC

☐ Don't optimize column width
☐ Return matching rows
☐ Trace MyODBC
☐ Allow BIG results
☐ Don't prompt on connect
☐ Simulate ODBC 1.0
☐ Ignore # in #.table
☐ Use manager cursors (exp)
☐ Don't use setlocale

☐ Pad CHAR to full length
☐ Return table names in SQLDescribeCol
☐ Use compressed protocol
☐ Ignore space after function names
☐ Force use of named pipes
☐ Change BIGINT columns to INT
☐ No catalog (exp)
☐ Read options from C:\my.cnf
☐ Safety (Check this if you have problems)
☐ Disable transactions

OK
Cancel

5. When the Query Wizard shows up, follow the instructions posted by the Wizard until finished.

Appendix C: Using the MOVES Summary Reporter

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

The Summary Reporter is accessed by selecting "Produce Summary Report" from the "Post-Processing" Menu:
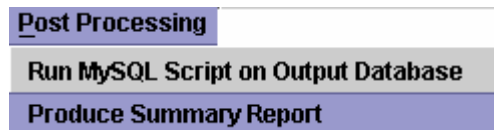

When selected, assuming a suitable run specification has been loaded, a popup appears asking the user to specify an emission process to be reported.
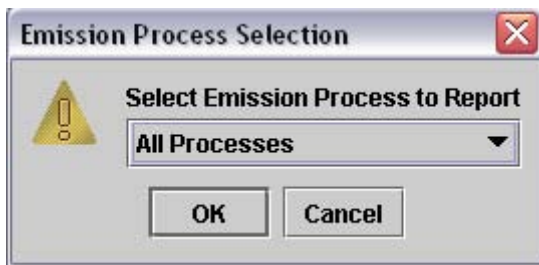
The default option (which is the only option if the run did not distinguish results by emission process) is to combine the output results for all emission processes included in the run

specification. But in many cases the user will want to select an emission process from the drop down list. In this example we will select the Evap Permeation process:
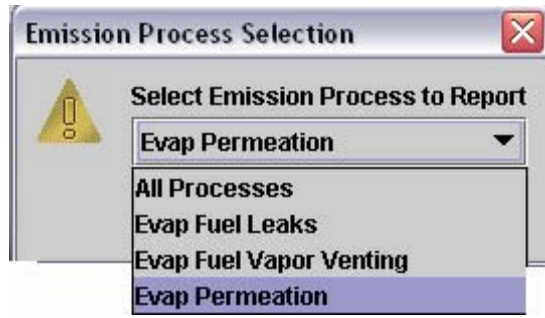
Clicking "OK" causes a popup to appear which allows the user to specify the content and level of aggregation of the report.
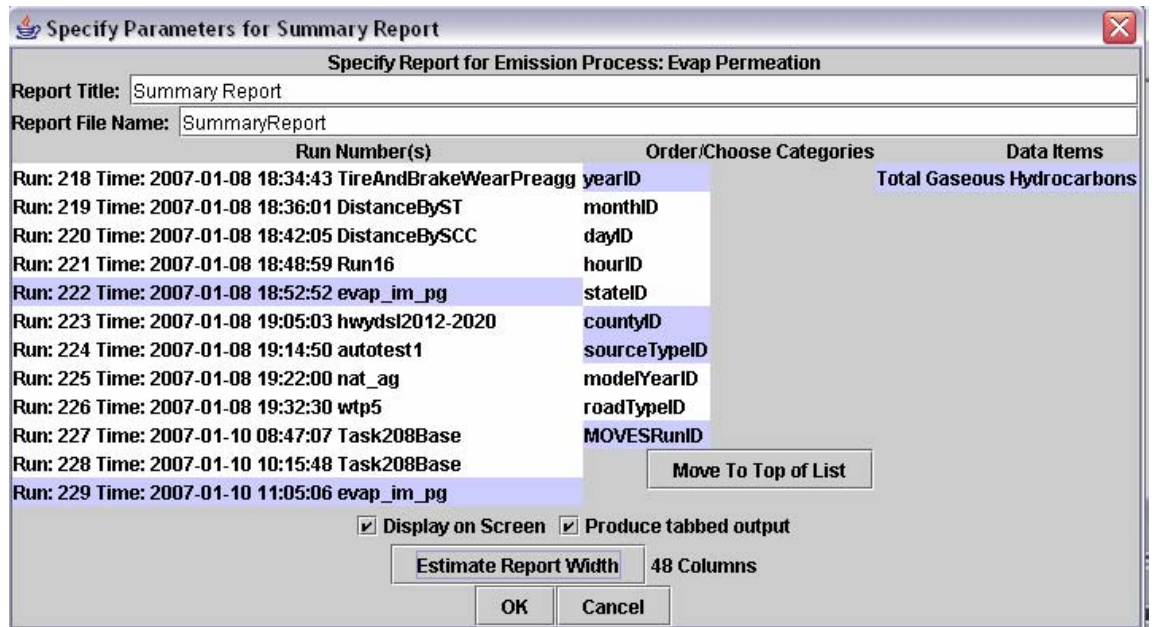
This screen allows the user to:

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

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

✒**CAUTION!** Files which already have this base file name will be overwritten without warning if reports are generated with the same base file name.

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

✒**CAUTION!** Care should be taken to compare runs with the same scope and level of output detail.

4. Choose the categories to be distinguished in the report. At least one category must be selected. The category list is based on the run specification. The fewer categories selected the more aggregate the report will be. Dimensions not distinguished when the run was made are not included. The "Move to Top of List" button may be used to reorder the category list.   The order of the list determines the sort order and the column order used in the report.   In this example several categories, including "MOVESRunID" have been selected.

!TIP If multiple runs are included in the report you will probably want to select the "MOVESRunID" category.   Otherwise

the results of the multiple runs will be combined in the report. Conversely if only a single run is included in the report, there is little reason to select the "MOVESRunID" category.

5. Select the data items to be included in the report. At least one data item, besides "distance" must be selected. In this example only one data item, "total gaseous hydrocarbons" is available, so it must be selected.

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

Because the width of a report can be a constraining factor, e.g. when printing the report, the "Estimate Report Width" button can be clicked. When this is done, as in the example screen above an estimate of the width of the report with the currently selected items appears in the popup window.

Printed report output is obtained by producing a screen report and then choosing to print it.

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

Screen Report

Year	County	Source	Run	TotalHC
2010	34021	11	222	10656
2010	34021	11	229	10656
2010	34021	21	222	564766
2010	34021	21	229	564766
2010	34021	31	222	217787
2010	34021	31	229	217787
2010	34021	32	222	66898
2010	34021	32	229	66898

Tabs: Header | Body | Decode

Buttons: Print and Close Report | Close Report

Note that the report consists of three portions. A Header, a Body, and a Decode page. The Body is shown initially, the other portions are accessed by clicking on their "tabs". The body of the report contains columns for each of the category and data items selected.

The header portion displays the report title, date and time the report was made, several items of information about the run specification, and the emission process covered by the report.

The Decode page portion decodes any numeric codes included in the report which are not self-explanatory or easily learned.
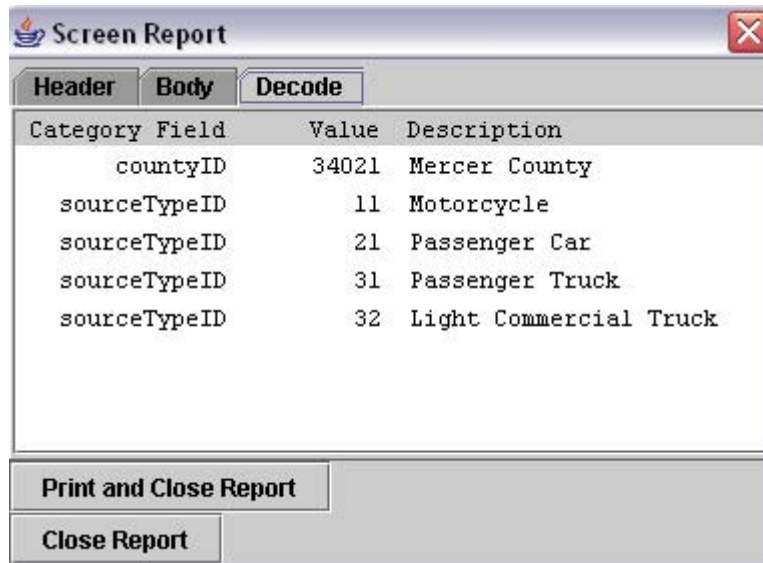
When the user is done viewing the screen report, they may click on "Print and Close Report" to print it. Printing is in landscape format. Clicking on "Close Report" closes the report without printing it.

✏**CAUTION!** Both buttons close the report. The Summary Reporter does not "remember" reporting options, other than the last report title and base file name used and these are not remembered between Summary Reporter runs, so do not close the report until you are really done viewing it.

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

Appendix D: Using the Batch Command Line Interface

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

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

java MOVESCommandLine -r *runspecfile*

[OR]

java MOVESCommandLine -rl *runspeclistfile*

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

What happens when you execute one of these commands is that you are running the java interpreter (java.exe). MOVESCommandLine is a DOS parameter telling java.exe what java class file to begin executing, and the last two tokens are parameters passed to MOVESCommandLine.

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

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

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

  the java interpreter
  the MOVESCommandLine java class
  any runspeclistfile and all runspecfiles

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

Running SETENV.BAT also insures that the procedure described in the next paragraph results in the MOVESCommandLine java class being found. (Java experts can

also use the CLASSPATH environment variable more directly to locate java class files.)

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

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