UNC
INSTITUTE FOR
THE ENVIRONMENT

Installation Guide for the SMOKE-MOVES Integration Tool

Contract No: EP-D-07-102
Work Assignment: 3-03

Prepared for:
Rich Mason
U.S. EPA, OAQPS
C339-02
USEPA Mailroom
Research Triangle Park, NC 27711

Prepared by:
B.H. Baek
Institute for the Environment
The University of North Carolina at Chapel Hill
137 E. Franklin St., CB 6116
Chapel Hill, NC 27599-6116

Date due: July 14, 2010

Contents

1 INTRODUCTION 1

2 INSTALLING AND PROCESSING STEPS 3

2.1 DOWNLOADING THE SMOKE-MOVES INTEGRATION TOOL 3

2.2 INSTALLING THE SMOKE-MOVES INTEGRATION TOOL 4

2.3 METEOROLOGICAL DATA PREPROCESSOR RUN SCRIPTS 5

2.3.1 Combining Meteorological Data: run_metcombine.MOVES.12km.csh 5

2.3.2 Averaging Meteorological Data: run_met4moves.MOVES.12km.csh 6

2.4 MOVES PROCESSING SCRIPTS 7

2.4.1 MOVES Driver Script: runspec_generator_v0.11.pl 8

2.4.2 MOVES Postprocessor Script: moves2smkEF_v0.13.pl 8

2.4.3 Running the MOVES driver script and postprocessor script 8

2.5 SMOKE RUN SCRIPTS 12

2.5.1 Modified SMOKE-ready Input Files for SMOKE-MOVES Integration Tool 14

2.5.2 SMOKE Run Scripts 14

2.5.3 Performing your own Processing 15

1 Introduction

The purpose of this manual is to describe the specific steps the user should follow to run the SMOKE-MOVES integration tool in order to develop emission rates based on lookup tables generated by the Motor Vehicle Emission Simulator (MOVES) and used by the Sparse Matrix Operator Kernel Emissions (SMOKE) model for regional-scale emissions modeling. This manual is a companion document to the "User's Guide for the SMOKE-MOVES Integration Tool." Detailed background information on the tool is provided in that user's guide, and so is not repeated here. The instructions in this document assume that you are familiar with the content of the user's guide.

Before we discuss the procedures for using the SMOKE-MOVES tool, it is important to make sure the user understands the concept of the "reference county," because the SMOKE-MOVES integration tool relies on this concept. Each reference county represents a set of similar counties (i.e., inventory counties) called a "county group." MOVES generates key emission rates for each reference county, and then SMOKE utilizes these factors to estimate emissions for all counties in the county group. All counties in a county group share the same fuel parameters, fleet age distribution, inspection/maintenance (I/M) programs, and similar meteorological conditions. The use of modeling emission factors for a reference county to represent an entire county group avoids unnecessary, duplicative MOVES runs. Each reference county is modeled at a range of speeds and temperatures to produce emission rate lookup tables (grams/mile or grams/vehicle/hour, depending on mobile emission process). This approach allows any county with unique distributions of vehicle miles traveled (VMT), vehicle population (VPOP), roadway speed, and grid cell temperatures to be modeled in SMOKE without having to rerun MOVES.

The two test cases provided as part of the SMOKE-MOVES integration package run two reference counties (i.e., 13121 and 13271) in Georgia for the month of April 2009. The package contains inventory files and ancillary input files needed by both the SMOKE and MOVES models. These include the annual VMT and dummy vehicle population inventory files (MBINV), the inventory table (INVTABLE), the gridding surrogates, the speciation profiles (GSPRO and GSPRO_COMBO), the mobile-source codes file (MCODE), the mobile reference county cross-reference file (MCXREF), the list of mobile emissions processes and associated pollutants (MEPROC) file, and others.

Note that these files are provided only as example inputs and should not be used for any modeling applications. For this test case, pre-generated lookup tables from the MOVES script (Section 2.4) are provided to be used to run the SMOKE post-processing script (Section 2.5). This allows the user to complete testing of Section 2.5 prior to the completion of Section 2.4 as the MOVES component takes 3-5 days per fuel month per reference county. The provided lookup tables were generated prior to the announcement of the bug in the MOVES post-processor and therefore the output generated with this release will not match the tables provided for testing the SMOKE post-processing script.

The SMOKE-MOVES integration package consists of the following components, listed in bullets that map to the three major facets of the tool described in detail in the SMOKE-MOVES integration tool user's guide:

- *Meteorological preprocessing:* The meteorological data preprocessor, **Met4moves**
- *MOVES model processing:* The MOVES driver script and the postprocessing script, both written in Perl
- *SMOKE model processing:* A version of the SMOKE modeling system to which the new SMOKE postprocessing program **Movesmrg** has been added

This document describes the steps necessary to use the run scripts from each processor, and is organized using the structure shown in the bullets above. The following is an overview of the process:

1. **Meteorological data preprocessor run scripts:**
   First of all, you will run the meteorological (met) preprocessor programs, Metcombine and Met4moves, to create meteorological input data for both the MOVES driver script and the SMOKE modeling system.

   a. ***Metcombine run script*** combines 2-D gridded met data with the first layer of 3-D gridded met data to create custom files for the met preprocessor Met4moves.

   b. ***Met4moves run script*** prepares spatially and temporally averaged temperature and relative humidity (RH) data to set up the meteorological input conditions for MOVES and SMOKE.

2. **MOVES processing scripts:**
   Once the meteorological input data have been processed and are ready for use, you will copy the output files from Met4moves to a Windows platform to execute the MOVES driver script. This script generates MOVES-based lookup tables and a list file, called MRCLIST that contains a list of lookup tables by fuel month that SMOKE uses to estimate emission rates by emission processes. The three processes are rate-per-distance (RPD), rate-per-vehicle (RPV) and rate-per-profile (RPP).

   a. ***MOVES driver script*** creates data importer files and the MOVES input file (runspec), which specifies the characteristics of the particular scenario to be modeled.

   b. ***MOVES postprocessing script*** reformats the emission rate lookup tables generated by MOVES and the MRCLIST file so that they can be used as input to SMOKE.

3. **SMOKE run scripts:**
   After the MOVES driver script and postprocessing script have generated all three of the MOVES-based lookup tables (RPD, RPV, and RPP) and the list file (MRCLIST), you will copy them to your Linux platform under the $GE_DAT/MOVES_lookuptables directory defined in the ASSIGNS file.

The MOVES emission processes are treated as individual sectors ($SECTOR) in SMOKE:

- Onroad sector: MOVES-based *rateperdistance* (RPD) lookup table
- Offroad-rpv sector: MOVES-based *ratepervehicle* (RPV) lookup table
- Offroad-rpp sector: MOVES-based *rateperprofile* (RPP) lookup table

You will copy MRCLIST input files to the location $INVDIR/$SECTOR directories for each sector ($SECTOR). You can also find detailed information about the format of MRCLIST file in the SMOKE-MOVES integration tool user's guide.

Once the MRCLIST files are copied, you will invoke the following three sector-specific run scripts to estimate emission factors using the MOVES-based lookup tables:

a. ***Onroad run script*** estimates on-roadway emission processes (e.g., exhaust running, on-road evaporative, tire and brake wear) based on the county-total VMT and average speed inventories, using the MOVES-based RPD lookup table.

b. ***Off-network vehicle run script*** estimates off-network emissions processes, except for the off-network fuel vapor venting emissions process, based on the county-total vehicle population inventory, using the MOVES-based RPV lookup table.

c. ***Off-network profile run script*** estimates only the off-network fuel vapor venting emissions process based on county-total vehicle population inventory, using the MOVES-based RPP lookup table.

2 Installing and Processing Steps

2.1 Downloading the SMOKE-MOVES Integration Tool

- Go to the CMAS Model Clearinghouse.
- If you have already registered with CMAS, enter your email address and password. Otherwise, click the "Register now" link and complete the registration form.
- You will be sent to the CMAS Download Center. Select SMOKE as the software family to download.
- Select "SMOKE-MOVES tool" in the pull-down option as the product you wish to download and select your computer and compiler type.
- There are several files including documentations, data, processing scripts and source codes. Save these files together on your machine according to the instructions.

- Save the "***MOVES_processing_scripts_14July2010.zip***" test case package on your local Window XP/Vista/7 machine for MOVES model. This package contains necessary sample input files and MOVES processing perl scripts that drive the MOVES model.

- Download the "installation script", "Source code, scripts, and executables", and "Test case data files" and save them in the same directory. The test case data package contains all SMOKE ancillary input data, dummy sample inventory data files, preprocessed dummy MOVES-based lookup tables, and a sample meteorology dataset.

2.2 Installing the SMOKE-MOVES Integration Tool

- Make sure that the three files you downloaded from the CMAS website are all in the same directory.

    1) ***smoke-moves_tool_install.csh***
    2) ***smoke.Linux2_x86pg.SMOKE-MOVES_tool.14July2010.tar***
    3) ***smoke.data.SMOKE-MOVES_tool.14July2010.tar***

- Begin the installation process by defining the SMK_HOME environment variable. Choose the directory where you want to install SMOKE and the test case packages. This directory will be known as your SMK_HOME directory. You can configure your UNIX system to set this automatically each time you log in to your computer. If you choose not to do that, then you will need to set the SMK_HOME variable manually. At the command line, type:

```
setenv SMK_HOME <your selected directory for SMOKE installation>
```

- Make sure your SMK_HOME directory exists by typing the following command on the command line: mkdir -p $SMK_HOME

- Run the installation script "*smoke-moves_tool_install.csh*". This script will automatically install SMOKE-MOVES integration tool under $SMK_HOME directory

- When using SMOKE, the first step is to go to the ASSIGNS directory and invoke the ASSIGNS file for the case that you will be working with. This makes available all of the environment variables that you can use to navigate through the SMOKE directory structure. There is a different ASSIGNS file for each case. In this manual, the instructions are specific to the ASSIGNS file associated with the test case package. When you later develop an ASSIGNS file for each of your own cases, you will use those files instead of the one shown below.

- To invoke the ASSIGNS file, use the following command:

```
cd $SMK_HOME/subsys/smoke/assigns
source ASSIGNS.MOVES_2009.cmaq.cb05p25tx.12km
```

If you see a message similar to "Do not know about executable format Linux2_x86_64ifort", it means that you need to update the $SMK_HOME/scripts/platform file to define $SMOKE_EXE

according to your platform. ***Note:*** the SMOKE scripts are designed to run under the UNIX C-shell (either csh or tcsh); the scripts will not work with other shells, such as bash.

- Move to $SCRIPTS/MOVES_2009.

```
cd $SCRIPTS/MOVES_2009
```

There are a total of five run scripts in the MOVES_2009 directory used to run **Metcombine**, **Met4moves,** and the three SMOKE run scripts for the three sectors (i.e., onroad, offroad-rpv and offroad-rpp). Detailed information about the sectors are described in Section 2.5.

To run the SMOKE-MOVES test case, you will invoke the following run scripts. These scripts and their use are discussed in Sections 2.3 and 2.5.

```
1) run_metcombine.MOVES.12km.csh
2) run_met4moves.MOVES.12km.csh
3) smk_onroad_MOVES_2009_12k.csh
4) smk_offroad-rpv_MOVES_2009_12k.csh
5) smk_offroad-rpp_MOVES_2009_12k.csh
```

2.3 Meteorological Data Preprocessor run scripts

2.3.1 Combining Meteorological Data: run_metcombine.MOVES.12km.csh

Metcombine is a SMOKE utility program that is used combine 2-D gridded met data with the first layer of 3-D gridded met data to create custom files for on-road mobile-source processing with MOVES, as follows:

- The pressure and mixing ratio data used in this step are typically found in the MET_CRO_3D MCIP meteorology data file with the variable names PRES and QV, respectively.
- Choosing the best temperature variable for computing relative humidity using mixing ratio and pressure for a MOVES run is up to the user. Possibilities include ambient air temperature (TA) from the METCRO_3D file, and ground temperature (TEMPG) or 2-m temperature (TEMP2) from the METCRO_2D file.

NOTE: If you decide to use ambient air temperature (TA) from the same METCRO_3D file that also contains PRES and QV, the running of Metcombine can be skipped.

NOTE: The sample METCRO_3D files provided in the test case package contain only the first-layer variables, in order to keep the file sizes manageable. The names of these files have been modified to reflect this change (e.g., METCRO3D_2009091.mod).

For more information on the **Metcombine** program, see the SMOKE user's manual (http://www.smoke-model.org/version2.6/html/ch05s10.html).

Running the Metcombine script:

The first step in running the **Metcombine** program is to invoke the ASSIGNS file (see Section 2.2). After you have done this, go to $SCRIPTS/MOVES_2009 and open the file run_metcombine.MOVES.12km.csh in an editor and review the following settings:

1. Start and end dates of the modeling period (`STDATE` and `ENDATE`, in Julian days)
2. The list of variables to combine together

   (e.g., `setenv VARLIST="TEMP2, PRES, QV"`)
3. The input meteorology file names (`METFILE1` and `METFILE2`), which are constructed from the case, grid, and date information
4. The combined output file name (`OUTFILE`), which is set based on case, grid, and date

Note: Please be aware that the script assumes that meteorological data files are available for each of the days you are modeling.

Once you have reviewed the $SCRIPTS/MOVES_2009/run_metcombine.MOVES.12km.csh, run the script. After the **Metcombine** program runs, the list of combined meteorological data files is stored to the METLIST file, which is defined in the ASSIGNS file. The METLIST file will be used by the **Met4moves** program later. The files can be seen in the directory used for the OUTFILE names: $MET_ROOT/MOVES_2009_12km/. The created file names start with METCOMBO (e.g., METCOMBO_2009091, METCOMBO_2009092).

2.3.2 Averaging Meteorological Data: run_met4moves.MOVES.12km.csh

Met4moves is a new program designed to create county-level averaged RH and minimum and maximum (min/max) grid cell temperatures for on-road mobile-source processing with the MOVES driver script. You must specify at least one spatial surrogate for selecting meteorology grid cells that represent modeling counties within the modeling domain. You can also specify more than one surrogate; the reason for using two different surrogates is to provide a proxy for grid cells with higher mobile emission activities. When **Met4moves** selects the absolute min/max temperatures for the reference county that it will feed to the MOVES driver script, **Met4moves** considers all of the selected grid cells in the reference county and all of the selected grid cells from all of the inventory counties assigned to the reference county. This approach is needed because the reference county could have a smaller temperature range than one or more of the counties that are mapped to it.

- For the MOVES driver script, **Met4moves** will create datasets that provide all min/max temperature combinations for a reference county, which will reflect all min/max combinations for all counties in that group. The associated RH values will also be included in these datasets. In addition, the datasets will include the 24-h diurnal profiles needed for the off-network fuel vapor-venting emission factors, and will also contain user-specified temperature increments for the *rateperdistance* (RPD), *ratepervehicle* (RPV), and *rateperprofile* (RPP) lookup tables that are used by the MOVES driver script. More detailed information is available in the SMOKE-MOVES integration tool user's guide.

- For the SMOKE modeling system, **Met4moves** will also create datasets that contain the min/max temperatures and averaged RH associated with each county for use in vapor-venting calculations performed in **Movesmrg** (discussed later).

CRITICAL NOTE: The list of counties in the MCXREF file used by Met4moves and the list of inventory counties used by SMOKE need to be exactly matched. You can only process the exact list of inventory counties listed in the MCXREF file through the SMOKE processing.

Running the Met4moves script:
Open the run script ($SCRIPTS/MOVES_2009/run_met4moves.MOVES.12km.csh) in an editor and review the following settings:

1. The start/end dates of the modeling period (`STDATE` and `ENDATE`, in Julian days).
2. One or more surrogate(s) that will be used to represent each modeling county, which are set by the `SRG_LIST` variable

   (e.g., `setenv SRG_LIST '100 240'`)
3. The name of the temperature variable from the `METCOMBO` data files

   (e.g., `setenv TVARNAME TEMP2`).
4. The averaging method for RH and temperature

   (e.g., `setenv AVERAGING_METHOD MONTHLY`).
5. The two output file names (`MOVES_OUTFILE` and `SMOKE_OUTFILE`), which are set based on the grid and the start and end dates.

As noted earlier, the METLIST file that contains the list of combined met files (METCOMBO) from the **Metcombine** run script is set by the ASSIGNS file. Also note that if you selected TA as the temperature variable for RH computation (see Section 2.3.1), the METLIST file should contain a list of METCRO_3D files instead of METCOMBO files.

Once you have reviewed the $SCRIPTS/MOVES_2009/run_met4moves.MOVES.12km.csh, run the script. After the **Met4moves** program runs, check the list of output files under the $MET_ROOT/MOVES_2009_12km directory.

The output files for the test case are titled "MOVES_12km_2009091-2009120.txt", which will be used by the MOVES driver script, and "SMOKE_12km_2009091-2009120.txt", which will be used by the SMOKE modeling system.

For more information on the **Met4moves** program, please see the SMOKE-MOVES integration tool user's guide.

2.4 MOVES Processing Scripts

Unlike the meteorology preprocessor and the SMOKE modeling system, the MOVES driver script is run on Windows XP/Vista/7. After you have finished running **Metcombine** and

Met4moves on Linux in the previous steps (Section 2.3), you must run the MOVES driver script and postprocessing script on your Windows system to prepare the MOVES-based lookup tables, before moving back to Linux to run SMOKE, using the lookup tables as input.

Once you download "**MOVES_processing_scripts_14July2010.zip**" from CMAS center, unzip it under the directory you designated for MOVES modeling. There are two perl scripts. The first one is MOVES driver script (i.e., **runspec_generator_v0.11.pl**) and the other is MOVES postprocessing script (i.e., **moves2smkEF_v0.13.pl**).

2.4.1 MOVES Driver Script: runspec_generator_v0.11.pl

The MOVES driver script generates the MOVES driver files—the run specifications (runspec) files and data importer files—based on the **Met4moves** output temperature list by reference county. The estimated run time on a Windows computer is a range of 3-5 days per reference county and fuel month (the run time will vary depending on processor speed). The script performs the following steps:

a) Reads the output file from **Met4moves** (Section 2.3.2) (***Note:*** the MOVES_12km_2009091-2009120.txt file created by **Met4moves** on Linux must be copied to your Windows machine before running the runspec generator program.)
b) Reads run control file and reference county file.
c) Assembles instructions for MOVES to create MySQL input databases from XML files (data importer).
d) Assembles runspec XML files to run MOVES for a wide range of conditions.
e) Generates the run-specific temperature and humidity comma-separated-values (CSV) file.
f) Assembles a batch list of (c) data importer files, (d) runspec files, and also a list of the MySQL output database names to be postprocessed.

2.4.2 MOVES Postprocessor Script: moves2smkEF_v0.13.pl

The MOVES postprocessor script extracts the emission factor tables from the MOVES databases; maps MOVES source, fuel, and road types to Source Classification Codes (SCCs); and formats the emission factor tables so that they can be used as SMOKE inputs. The estimated run time on a Windows computer is between 1-2 days per reference county and fuel month. To accomplish this, the script performs the following steps:

a) Reads a list of databases created by the MOVES driver script.
b) Performs mapping from MOVES source types, fuel types, and road types to SCC, and aggregates SCC emission factors over model years.
c) Performs operations to drop and add fields and reduce database size by performing a cross-tab query that places all pollutants on a single row.
d) Writes final processed lookup tables to CSV files formatted for SMOKE.

2.4.3 Running the MOVES driver script and postprocessor script

2.4.3.1 Preliminary steps:

- Download and install ActivePerl for Windows.

- Enable ActivePerl for Windows to talk to MySQL, as follows:
  - Open the Perl Package Manager (PPM) from the Windows Start Menu > Programs.
  - On the menu bar of the PPM, select View > All Packages.
  - In the search bar type *DBD-mysql* and select the package when it appears in the list. On the menu bar, select Action>Install DBD-mysql, and then select File>Run Marked Options.
  - **Important: Reboot your machine after installing ActivePerl to complete the installation.**

2.4.3.2 Running the scripts:

1. Prepare all inputs for the MOVES driver script, including
   - Run control file: "control.in"
   - Reference county file: "countyrep.in" (see item 1.1 below)
   - **Met4moves** output for MOVES driver script: "MOVES_12km_2009091-2009120.txt"
   - MOVES inputs following the County Data Manager templates in CSV format for each reference county:
     - sourceTypeAgeDistribution (e.g., "agedistribution_13121.csv")
     - fuelSupply (e.g., "fuelsupply_13121.csv")
     - fuelFormulation (optional)
     - IMcoverage (e.g., "imcoverage_13121.csv")
     - sourceTypeYear[1](e.g., "population_13121.csv")
     - HPMSVTypeYear[2] (e.g., "hpmsvtypeyear_13121.csv")
   - Place additional MOVES "dummy" input files provided with the tool package in the directory specified as OUTDIR in the Run Control File (see item 1.2 below). These include:
     - "dummy_avgspeeddistribution.csv"
     - "dummy_monthvmtfraction.csv"
     - "dummy_dayvmtfraction.csv"
     - "dummy_hourvmtfraction.csv"
     - "dummy_roadtypedistribution.csv"

1.1. Reference County File : "countyrep.in"

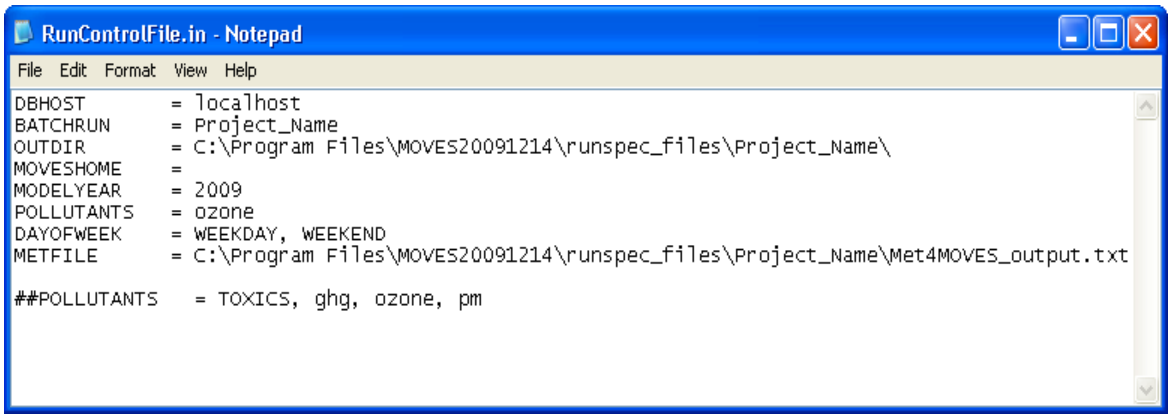
List reference counties as separate REPCOUNTY packets, as shown below. Each reference county needs entries for the FIPS field and full path and file names for AGE, IM, FUELSUPPLY, POP, HPMSVMT, and FUELFORM (optional). Every county specified in the **Met4moves** output file must be specified in the reference county file or an error will occur.

[1] Per updated MOVES2010 guidance released April 2010 (Section 3.3.1): Source type population inputs must cover the entire area over which the emission rates are to apply. For SMOKE application, a reference county represents a group of counties and so population inputs must be prepared at the appropriate geographic resolution.

[2] Per updated MOVES2010 guidance released April 2010 (Section 3.5.1): HPMSvType VMT inputs must cover the entire area over which the emission rates are to apply. For SMOKE application, a reference county represents a group of counties and so VMT inputs must be prepared at the appropriate geographic resolution.

1.1. Run Control File "control.in"

Create the run control file specifying fields for DBHOST, BATCHRUN, OUTDIR, MOVESHOME (optional), MODELYEAR, POLLUTANTS, DAYOFWEEK, and METFILE. Each keyword is followed by an equal sign ("=") and the user's chosen option.



2. Open the driver script (runspec_generator_v0.1.pl) in a text editor (e.g., Notepad). You may need to add a ".txt" extension to the end of the file name for Windows to recognize it as a text file. Use the search function in the text editor to locate the "$moveshome" environment variable that points to the top-level MOVES program directory. Change the path to reflect the location of MOVES on your machine. When you are finished, remember to remove the ".txt" extension from the file name if you added it.

3. Run the driver script. First, open a DOS command prompt window (Start > Programs > Accessories > Command Prompt). Change directories to the current working directory and execute the following command:

```
>perl runspec_generator_v0.11.pl RunControlFile.in RefCountyFile.in
```

This step generates the MOVES-formatted zonemonthhour CSV files (*_zmh.csv), data importer files (*_imp.xml), and runspec files (*_mrs.xml), writing all files to the location specified with the OUTDIR keyword in the run control file.

This step also generates two batch file lists (written to OUTDIR), one for data import to MySQL (*importer.bat, step 4 below) and one for running MOVES for each runspec (*runspec.bat, step 5 below).

4. Run the importer batch file to create the MySQL MOVES databases and import the external datafiles. Execute the importer batch file (*importer.bat) by typing the batch file's full name in the DOS command prompt window. When the import completes, check the importer log (importlog_*.txt) for errors and review the MySQL databases created (*_in) if desired.

5. Execute MOVES runs. Execute the runspec batch file (*runspec.bat) by typing the batch file's full name in the DOS command prompt window.
Highly recommended tip: **Before executing** the runspec batch file, open the runspec batch file in a text editor and perform a search-and-replace on "java gov" to order to **allocate additional computer memory** to the java application. Specifically, replace "java gov" with "java –XmxNNNm gov" where NNN is an integer (does not have to be three digits). The maximum amount of memory available for the java application varies from computer to computer. Refer to Appendix C of the MOVES2010 user's guide.

6. Wait for MOVES runs to complete. Depending on the size of RAM memory and CPU speed in your machine, the computational time may vary. But it will likely take several days to complete the MOVES runs.

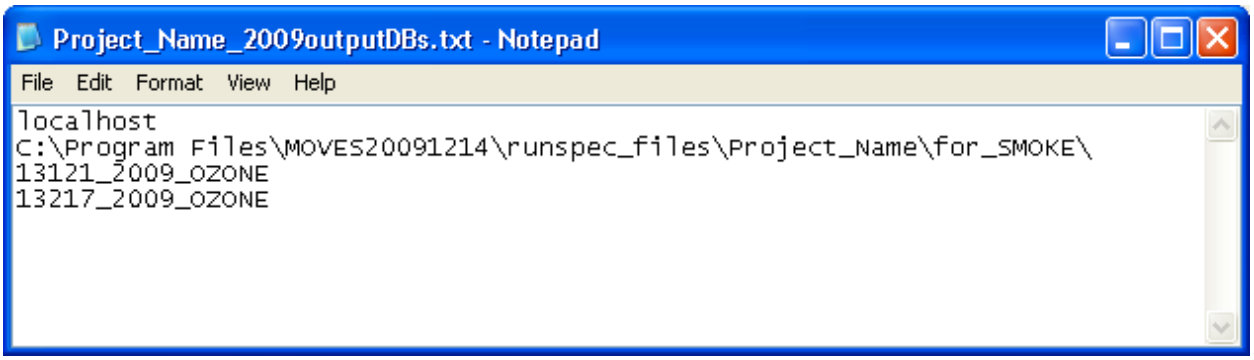
7. Review the MOVES run log that has been automatically created, called *runlog_*.txt,* and search for "ERROR" to be sure the run completed without problems.

8. Run the postprocessor script by opening a DOS command window and entering the following command.

```
>perl moves2smkEF_v0.13pl *outputDBs.txt
```

Note: If you encounter the error: "*The total number of locks exceeds the lock table size at moves2smkEF_v0.13.pl line 478*", then edit the my.ini file located under the MySQL directory to increase the buffer pool size (innodb_buffer_pool_size) to greater than or equal to 18MB.

The file *outputDBs.txt is automatically created by the driver script runspec_generator.pl. It lists the MySQL server name, the directory where the SMOKE-formatted final lookup tables will be written, and the MOVES output database names on separate lines that are to be formatted for SMOKE. An example *outputDBs.txt file is shown below.



9. Wait for the MOVES postprocessor script to complete. Because many calculations are being performed within MySQL, this step will likely take several hours.

10. The end product will be fuel month-specific three lookup tables for each reference county in CSV format in the directory specified in *outputDBs.txt on line 2. Also there will be a list text output files (MRCLIST) that contain a list of these lookup table names by fuel month for SMOKE to use as an input file in Section 2.5. You can find more information from SMOKE-MOVES integration tool user's guide.

    a. rateperdistance_smoke_*.csv
    b. ratepervehicle_smoke_*.csv
    c. rateperprofile_smoke_*.csv

2.5 SMOKE Run Scripts

Once the SMOKE-formatted emissions factor lookup tables from the MOVES driver script and postprocessor script have been finalized, you will use the final component of the SMOKE-MOVES integration tool: the SMOKE system, including the SMOKE postprocessing program **Movesmrg.** This section will show you how to process the various MOVES-based lookup tables through the SMOKE modeling system.

Note: For this test run, you will not use the lookup tables generated by running the MOVES perl scripts on Windows in the above section (Section 2.4). Use the lookup tables provided in this test case under the $GE_DAT/MOVES_lookuptables directory for the SMOKE runs. There is an inconsistency in Met4moves input file between the MOVES scripts provided in this release and the test case. This inconsistency is due to fixes of the bug reported on July 6th, 2010 related to how PM species were computed in MOVES post-processing script.

For your own production runs, copy the files output from the moves2smkEF script (Section 2.4.2) into the $GE_DAT/MOVES_lookuptables directory on the Linux computer that will be used to run SMOKE. The lookup files names start with rateperdistance_smoke, rateperprofile_smoke, and ratepervehicle_smoke**,** and the text output files (MRCLIST) that contain a list of these lookup table names by fuel month.

Once you complete processing SMOKE runs, you will check your output reports from SMOKE runs against the QA reports to confirm that your installation of SMOKE is complete.

Once the county-total VMT data and vehicle population (VPOP) data by SCC are imported into SMOKE (via Smkinven), the chemical speciation step (Spcmat) converts VMT and VPOP into the chemical speciation factors for each county, source type, road type, emission process, and pollutant. The gridding step (Grdmat) allocates the county sources to grid cells and uses spatial surrogates to allocate county-total VMT and VPOP to grid cells, storing the factors needed for these allocations. For more information on using the previously existing parts of SMOKE for MOVES data, see the SMOKE-MOVES integration tool user's guide.
CRITICAL NOTE: Only counties listed in the MCXREF file can be processed through the SMOKE-MOVES tool.

Movesmrg is loosely based on the SMOKE program Smkmerge, which combines the intermediate files produced by the other SMOKE programs (i.e., Grmdat, Spcmat, Temporal). Similar to Smkmerge, the main goal of **Movesmrg** is to create (1) gridded, speciated, hourly air quality model-ready input files and (2) county-level SCC summaries, using the SMOKE-formatted emission factor lookup tables from MOVES. The primary difference between Movesmrg and Smkmerge is that Movesmrg uses the MOVES emission rate lookup tables to compute emissions on the fly, by multiplying the emissions factors by hourly VMT (for RPD, which has grams/mile factors) or by monthly or annual vehicle populations (for RPV or RPP, which have for grams/vehicle/hour factors).

The way the MOVES-generated emissions factor lookup tables are used varies according to whether SMOKE is modeling on-roadway emission processes or off-network emissions processes.

On-roadway emission processes: When estimates of on-roadway emission processes (e.g., exhaust running, on-road evaporative, tire and brake wear) are needed, SMOKE requires county-total VMT and average-speed (SPEED) inventory data as inputs to **Movesmrg** to use the *rateperdistance* (RPD) table. The key lookup fields for the factors are gridded hourly temperature and average hourly speed. SMOKE interpolates in the RPD table (in units of grams/mile) based on gridded hourly temperature and average speed.

Off-network emission processes: When estimates are needed for off-network emissions processes, including the off-network vapor venting emissions process, SMOKE uses county-total VPOP by vehicle type as input to **Movesmrg** together with the *ratepervehicle* (RPV) and *rateperprofile* (RPP) lookup tables. A significant difference in the processing steps between the on-roadway emissions processes (RPD table) and the off-network emissions processes (RPV and RPP tables) is that off-network emissions processing does not require the Temporal step because vehicle population (VPOP) does not need to be temporally allocated. In the RPV table, gridded hourly temperature and hour of the day are the key lookup fields SMOKE uses to estimate hourly off-network emissions in unit of grams/vehicle/hour. For the evaporative fuel off-network vapor venting emissions process, **Movesmrg** uses the RPP lookup table to estimate the emission rates based on the minimum and maximum temperatures computed by Met4moves.

Movesmrg must be run three times to compute on-roadway and off-network (including vapor venting) emissions processes individually, since the emission rate calculation methods in SMOKE for these categories are quite different. The input environment variables used by Movesmrg are described below. Prior to running the scripts, be sure that the METMOVES environment variable in the ASSIGNS file is set to the name of the SMOKE_ file created by **Met4moves.**

2.5.1 Modified SMOKE-ready Input Files for SMOKE-MOVES Integration Tool

There are several SMOKE-ready input files in the test case package that have been modified to support the processing of the MOVES-based lookup tables through SMOKE. The modifications are required due to the creation of new MOVES emission processIDs (=smokeProcIDs) (e.g., CXR, CXS). See Section 4 in the SMOKE-MOVES integration tool user's guide for more detailed information. Here is a list of the modified SMOKE-ready input files that are needed to support these new emissions smokeProcIDs:.

- MBINV: VMT and dummy VPOP inventory files
- GSREF: Chemical speciation cross-reference file
- GSPRO_COMBO: County-specific speciation combo profile file
- MEPROC: File containing a list of MOVES smokeProcIDs and associated activities and pollutants

NOTE: These files are provided only as example inputs and should not be used for any modeling applications.

2.5.2 SMOKE Run Scripts

After the MOVES driver script has generates all three MOVES-based lookup tables (RPD, RPV and RPP), copy them to your Linux platform under $GE_DAT/MOVES_lookuptables. The MOVES emission processes are treated as individual sectors ($SECTOR) in SMOKE:

- Onroad sector: MOVES-based *rateperdistance* (RPD) lookup table
- Offroad-rpv sector: MOVES-based *ratepervehicle* (RPV) lookup table
- Offroad-rpp sector: MOVES-based *rateperprofile* (RPP) lookup table

Once you complete copying the lookup tables from Section 2.4.2, copy the sector-specific lookup tables list files (MRCLIST) under $INVDIR/$SECTOR directory and invoke the SMOKE run scripts to process the MOVES-based lookup tables separately for the test case, which is set up to run the month of April (-m 4) 2009.

NOTE: Each of these scripts will run the various SMOKE programs (i.e., Smkinven, Spcmat, Grdmat, Temporal, Movesmrg, and Smkreport); the set of programs that are run in each case will depend on the sector involved.

1. Onroad sector: MOVES-based RPD lookup table

```
> smk_onroad_MOVES_2009b_12k.csh –m 4
```

2. Offroad-rpv sector: MOVES-based RPV lookup table

```
> smk_offroad-rpv_MOVES_2009b_12k.csh –m 4
```

3. Offroad-rpp sector: MOVES-based RPP lookup table

```
> smk_offroad-rpp_MOVES_2009b_12k.csh –m 4
```

Once the scripts finish running, move to $SMK_HOME/intermed/MOVES_2009 to see the output files in the appropriate directories: offroad-rpp, offroad-rpv, and onroad. The gridded, speciated, hourly emissions file names start with emis_mole. Note that the offroad-rpp sector is processed for only one representative (average) day per month, while data for the other two sectors are produced for every day of each month. After emissions from all three emissions processes have been created, the resulting files can be merged (optionally along with other 2-D sectors separate from onroad) using Mrggrid, which combines any number of 2-D or 3-D hourly, gridded, and speciated emission data files. For more information on the settings for **Mrggrid**, see the SMOKE user's manual at http://www.smoke-model.org/version2.6/html/ch06s10.html.

To make sure everything worked properly as you worked through the steps in this manual, you can compare the county-total report provided in the test case with the one you created. The test case county-total report is located at $REPOUT/MOVES_2009/answers/$SECTOR; your county-total report is located at $REPOUT/MOVES_2009/movesmrg/$SECTOR.

2.5.3 Performing your own Processing

Once you have successfully replicated the test case, you can move on to applying the SMOKE-MOVES integration tool using your own data. Before running SMOKE-MOVES for your own case, however, you must create/modify the input files listed in Section 2.5.1 to make them appropriate for your modeling case.