Instructions for running an automated water-use permit simulation

Most recent update to this document: 2018-01-18

Disclaimer and Request for Error Reports:

Please report any errors or corrections that otherwise need to be made to the Suwannee River Water Management District:

email: hydrologic_modeling@srwmd.org

phone: 386.362.1001

mail: Suwannee River Water Management District
9225 CR 49
Live Oak FL 32060

Software Requirements

ArcGIS Desktop, version 10.0 or later, installed on your local machine (or a computer that you connect with remotely that has ArcGIS installed on it).

Setup/Installation (only needs to be done once)

1. Open a file browser (Windows Explorer, for example), and create a directory called ‘cup_sims_nfmv2’ under your user directory on your local hard drive (e.g. if your ‘home directory’ is called, c:\users\jane_smith, then you need to create a directory called, c:\users\jane_smith\cup_sims_nfmv2).

2. Copy and unzip the file, wup_sim.zip[1], to this newly-created directory. At this point you should have two subdirectories, .\cup_sims_nfmv2\archive and .\cup_sims_nfmv2\working under your user directory
3. If you have a copy of the ArcGIS software installed on your machine, then you should also have a copy of the Python programming language (version 2.7) that was installed for use with ArcGIS. Make a note of the directory (folder) that ArcGIS installed the Python programming language interpreter into by searching for the file, python.exe on your machine. It will probably be located on a directory path like c:\Python27\ArcGIS10.2.
4. If ArcGIS is not installed in the location, c:\Python27\ArcGIS10.2, then open the file, sim_cup.bat, in a text editor (e.g. Notepad, UltraEdit, Notepad++, Vim, Emacs, etc.), and globally search and replace the string, 'c:\Python27\ArcGIS10.2' (without the quotes), with the corresponding path on your PC.

Executing a permit simulation

1. If this isn't the first time you've done a permit simulation, then delete the directory structure that you created during your initial setup/installation (see previous section) or any subsequent permit simulation. Once this is done, reexecute steps 2 through 4 (as necessary) described in the previous section, Setup/Installation.
2. Open up a command/console window and change to the working directory. This can be done in a variety of ways. Here are two examples:
    a. Open the Windows Explorer file browser (right-click on the Windows start button and select Explorer), browse to the working directory (e.g. c:\users\jane_smith\cup_sims\nfmv2\working), then type, 'cmd.exe' (without the quotes) in the path/location bar of Windows Explorer.
    b. Click on the Windows start button, type 'cmd.exe' (without the quotes) in the search box, click on the cmd.exe icon, then use the 'cd' command to change to the working directory (e.g. type '`cd \users\jane_smith\cup_sims_nfmv2\working`' (without the quotes) at the command prompt in the cmd.exe console window.
3. Open the file, sim_cup_input.xlsx, in Microsoft Excel and edit as follows:
    a. Input the correct permit number and name in cells, A1 and A2, respectively. If necessary, please 'simplify' the permit number and name by removing any spaces or commas from the permit number or name. It's also suggested (but not strictly required) to replace dashes with underscores (this often makes data processing cleaner or easier to perform if you want to do additional postprocessing outside of what is performed by the 'batch file', sim_cup.bat).I like to use a combination of 'camel case' and underscores (to replace spaces) when I simplify the permit numbers and names. For example the permit number, 2-041-220604-3 would become, 2_041_220604_3, and the permit name, American DairyCo – Bell, would become, AmericanDairyCo_Bell.
    b. Leave row 2 unchanged – it is just a list of column names for well data that are stored in rows 3 and higher.

[1] On the SRWMD computer network, this file can be found at \\POSEIDON\WaterUseReporting2007\Water Use Permit Issuance\nfmv2_model_archive\wup_sim.zip

c. Each row in rows 3 and higher contains data associated with an individual well in the permit. There should be one row for each permit, unless a well withdraws water from more than one model layer (see Appendix A for details on inputting data for wells withdrawing water from more than one well). The following information is input for a given well:
    i. Column A: numeric identifier of the well
    ii. Column B: name of the well
    iii. Column C[2]: x-coordinate (east-west coordinate) for the well's horizontal location in the Florida State Plane North projection system (see Appendix B for projection details)
    iv. Column D[2]: y-coordinate (north-south coordinate) for the well's horizontal location in the Florida State Plane North projection system (see Appendix B for projection details)
    v. Column E: model layer from which the withdrawal is occurring.
    vi. Column F: withdrawal rate from the well, in millions of gallon per day
d. Save the Excel file in its native .xlsx format.
e. Save the file as the .csv-formatted file, sim_cup_input.csv (i.e. as a plain-text, comma-delimited file)
f. Exit Excel

4. Execute the model-simulation 'batch file' by typing the following at the command line of the cmd.exe console window[3]:

```
sim_cup.bat > sim_cup.log 2>&1
```

Note that it takes several minutes for the batch file to finish executing. Once it has completed, the command prompt (e.g. c:\users\jane_smith\cup_sims\nfmv2\working>) will reappear.

This one command will do the following:

a. Determine the location of the permitted wells in the model and create a new MODFLOW Well Package input file with two stress periods (the first stress period representing withdrawals from calendar-year 1995, and the second period with the 1995 withdrawals plus those from the permit being evaluated).
b. Execute the simulation with two steady-state stress periods. The only difference between the simulations in these two stress periods is the addition of the permit wells to the stress period 2 simulation (as described in the previous step).
c. Postprocess the simulation results as follows:
    i. Evaluate the change in the simulated water budget from stress period 1 to stress period 2 over the entire model domain, and generate a corresponding report.

[2] These coordinates can be in the UTM zone 17 N (length units=meters) coordinate system as long as the batch file, sim_cup_sjr.bat is called instead of sim_cup.bat (see step 4 of section, Executing a permit simulation).
[3] Execute the command, sim_cup_sjr.bat > sim_cup.log 2>&1, if you entered x, y coordinates in the UTM Zone 17N projection system in columns C and D of file, sim_cup_input.xlsx (see step 3.c in section,Executing a permit simulation)

ii. Calculate simulated exchanges of water between the groundwater flow system represented in the model and selected river reaches and springs, and the changes in these simulated fluxes from stress period 1 to stress period 2.
iii. Calculate simulated changes in groundwater levels from stress period 1 to stress period 2 for model layers 1 and 2, and generate corresponding gis feature classes and an ArcGIS map document (mxd file) displaying those differences and the locations of wells in the permit.
d. Log output from the console window to a file called, sim_cup.log.

5. Create a zip archive of the working directory. It is recommended that the naming convention, 'gw_model_results_x_xxx_xxxxxx_x.zip' be used to name the archive (where x_xxx_xxxxxx_x represents the permit id number. For example, the zip archive for a permit with id, 2-041-220604-3, would be, gw_model_results_2_041_220604_3.zip.
6. Evaluate the modeling results in the project directory for the permit (in the working directory that was created when you unzipped the gw-modeling results file ('gw_model_results_x_xxx_xxxxxx_x.zip') in step 6, as described in the Evaluation of Results section (along with any other checks and evaluations of the modeling results not described in this section that you may also want/need to do).
7. Copy the zip file to your project directory for this particular permit so that it can be archived along with the other digital files associated with the permit. Unzip the file in your project directory.
8. Delete the directory, .\working (e.g. c:\users\jane_smith\cup_sims_nfmv2\working), on your local hard drive so that this space will be free to work in for your next permit.

Evaluation of Results[4]

1. The file, delta_q_summary.csv, in the directory, .\working, contains a table with simulated exchanges of water between the groundwater flow system represented in the model and selected river reaches and springs, and the changes in these simulated fluxes from stress period 1 to stress period 2. The easiest way to view the contents of this file is to double (left mouse button) click on it. This should open it up in Microsoft Excel. This file contains the following columns of data:
   a. ***station_number***: Identification number of the location of the simulated fluxes (for springs) or downstream limit of a given set of river reaches (typically occurs at the location of a stream-gaging station).

[4] These steps are general suggestions for evaluating simulations of changes in groundwater-surfacewater exchanges in response to withdrawals from wells. They are not intended to cover all circumstances, nor are they intended to describe all of the quality-control/assurance and checking that you may wish to conduct when simulating the effects of withdrawals. Please feel free to supplement them with any additional checking and analyses that you may want to carry out, or even to use a different model and pre- and postprocessing approaches. Above all, apply a healthy dose of common sense and professional judgement when evaluating, interpreting, and applying any simulation results. The software used to carry out the simulations described in this document should be used at your own risk. Please report any errors to the Suwannee River Water Management District (jwg@srwmd.org; see previous section, Disclaimer and Request for Error Reports)..

b. ***station_name*** : name of the location of the simulated fluxes (for springs) or downstream limit of a given set of river reaches (typically occurs at the location of a stream-gaging station).
c. ***simulated_flux_base_condition_cfs***: Simulated exchange (flow) between the groundwater flow system and the river reach or spring associated with this row (identified by values in the previous two columns of this row) in cubic feet per second (cfs) under the ‘base’ condition (1995 withdrawals without the additional withdrawal data specified in the file, sim_cup_input.csv). Note that negative values indicate that the model simulated a net flow *from* an aquifer (or multiple aquifers) *into* the river reach or spring. Also note that, for river reaches, the value represents the accumulation of simulated flows between the groundwater flow system and all of the river reaches in the model that flow toward the station specified by the ***station_number*** and ***station_name*** values for a particular row in the table.
d. ***simulated_flux_with_cup_cfs***: Simulated exchange (flow) between the groundwater flow system and the river reach or spring associated with this row (identified by values in the previous two columns of this row) in cubic feet per second (cfs) under the ‘stress period 2’ condition (1995 withdrawals ***plus*** the additional withdrawal data specified in the file, sim_cup_input.csv). Note that negative values indicate that the model simulated a net flow *from* an aquifer (or multiple aquifers) *into* the river reach or spring. Also note that, for river reaches, the value represents the accumulation of simulated flows between the groundwater flow system and all of the river reaches in the model that flow toward the station specified by the ***station_number*** and ***station_name*** values for a particular row in the table.
e. ***simulated_change_in_flow***: Difference between ***simulated_flux_with_cup_cfs*** and ***simulated_flux_base_condition_cfs*** (i.e. the value, ***simulated_flux_with_cup_cfs*** - ***simulated_flux_base_condition_cfs)***. Note that positive values indicate a simulated decline in the amount of water flowing from the groundwater flow system to a particular surface-water feature (e.g. river reach or spring).
f. ***simulated_percent_change_in_flow***: This column is computed as follows:

$$100 * simulated\_change\_in\_flow \;/\; simulated\_flux\_base\_condition\_cfs$$

Note that if the value of ***simulated_flux_base_condition_cfs*** is close to zero, then a value of -1.2345e+30 will be assigned.

g. ***simulated_change_in_flow_as_a_fraction_of_cup:*** This column is computed by dividing the value of ***simulated_change_in_flow*** by the total withdrawals specified among all of the wells in rows 3 and higher of file, sim_cup_input.csv. This number should also be less than 1 (within the precision of the model solution, which will probably be no smaller than 0.001 cfs).

2. The file, global_budget.csv, contains a table with information on the simulated water budget for the model domain for stress periods 1 and 2, and the change in those simulated budgets between the two stress periods. This table is produced for quality assurance (and to better understand the simulation). The table contains the simulated fluxes for each of the boundary condition types in the model. The simulated flux data are repeated three times in the table,

differing only in the units associated with the flux values reported in columns 7 through 15. The data contained in this file are as follows:

a. Column 1 (bc_flux_type): type of MODFLOW boundary condition
b. Column 2 (flux_units): units associated with simulated flux values reported in columns 7-15:
    i. 'cfd' = cubic feet per day
    ii. 'cfs' = cubic feet per second
    iii. 'mgd' = millions of gallons per day
c. Columns 3 and 4 are the MODFLOW time step and stress period numbers for the first stress period (not very useful for this particular application)
d. Columns 5 and 6 are the MODFLOW time step and stress period numbers for the second stress period (not very useful for this particular application)
e. Columns 7-9 are simulated inflows for the entire model domain for a given MODFLOW boundary condition type. Specifically:
    i. Column 7 (in_rate_1) is the inflow for a given boundary-condition type for stress period 1
    ii. Column 8 (in_rate_2) is the inflow for a given boundary-condition type for stress period 2
    iii. Column 9 (in_rate_2_minus_1) is equal to Column 8 minus Column 7
f. Columns 10-12 are simulated outflows for the entire model domain for a given MODFLOW boundary condition type. Specifically:
    i. Column 10 (out_rate_1) is the outflow for a given boundary-condition type for stress period 1
    ii. Column 11 (out_rate_2) is the outflow for a given boundary-condition type for stress period 2
    iii. Column 12 (out_rate_2_minus_1) is equal to Column 11 minus Column 10
g. Columns 13-15 are simulated net flows (inflow minus outflow) for the entire model domain for a given MODFLOW boundary condition type. Specifically:
    i. Column 13 (net_rate_1) is the net flow for a given boundary-condition type for stress period 1
    ii. Column 14 (net_rate_2) is the net flow for a given boundary-condition type for stress period 2
    iii. Column 15 (net_rate_2_minus_1) is equal to Column 14 minus Column 13
h. Column 16 (net_rate_2_minus_1_fraction_of_netWellPkg) is equal to column 15 divided by the sum of all of the withdrawals specified for the permit in the file, sim_cup_input.csv. It is a very useful way of expressing the changes in the individual boundary-condition fluxes between the two stress periods. For example, if a given row has value of 'RIVER LEAKAGE' for column 1 and a value of 0.23 for column 16, this indicates that the change in the simulated exchange between the groundwater flow system and the River Package boundary condition represents 23 percent of the sum of the additional well withdrawals in stress period 2.

    The file, global_budget_change.csv, can be used to provide a number of other quality assurance checks. For example, When the value of column 1 is 'WELLS', then the value of column 16 should be equal to 1, and the value of column 15 should be equal to the sum of the additional well withdrawals specified in the input file, sim_cup_input.csv. If these numbers don't match up (within the precision of the simulation results), it could be an indication that locations of the wells were incorrectly specified in file, sim_cup_input.csv. It could also occur (for example) if a well has been placed in a 'constant head' cell, which might occur for shallow (layer 1) wells along the coastline.

3. Maps of simulated changes in groundwater levels can be found by opening the ArcGIS map document file, dh.mxd, in the directory, .\working\gis. Simulated changes for model layers 1 and 2 are represented with GIS layers, dh_layer_1 and dh_layer_2, respectively. Symbology can be imported from any of the following layer files[5]:
    i. dh_minus02_to_10.lyr (lower limits of 'bins' range from -10^{-2} to less than -10 feet)
    ii. dh_minus03_to_1.lyr (lower limits of 'bins' range from -10^{-3} to less than -1 foot)
    iii. dh_minus04_to_ minus03.lyr (lower limits of 'bins' range from -10^{-4} to less than -10^{-1} foot)
    iv. dh_minus05_to_ minus02.lyr (lower limits of 'bins' range from -10^{-5} to less than -10^{-2} foot)

   Note that these simulated head changes should be interpreted with caution because they depend on transmissivity estimates from a regional groundwater flow model. Simulated drawdowns will differ from the actual drawdown because of differences between modeled and actual values of system hydraulic properties, stresses, etc. Differences will also occur because of model resolution and proximity of withdrawals to certain types of model boundaries. If you need to make site-specific estimates of drawdown, you might consider using a local model (or an analysis based on an analytical solution to a simplified representation of the system) with a range of plausible values of hydraulic properties based on knowledge of the local hydrogeology.

Appendix A: Creation of file, sim_cup_input.csv, for multi-layer wells

If a well is open to more than one model layer, then withdrawals from this well are represented in the file, sim_cup_input.csv, with multiple rows (instead of 1 row). Each of the rows that are being used to represent this multilayer well have identical information for all but the last (rightmost) two columns. These last two columns will contain the model layer and withdrawal rate for the well from the model layer specified on this second to the last column (column E if viewed in Microsoft Excel) will contain the model layer and corresponding withdrawal rate from that model layer by the multilayer well. The withdrawal rates from each model layer can be calculated as follows:

1. For each model layer, determine the length of open hole or well screen in the model layer, and multiply this length by the hydraulic conductivity of the layer in

[5] If you import one of these layer files, please perform the following to make sure that ArcGIS resets the lower limit of the lowest symbol classification (this is a 'quirk' in ArcGIS): click the Classify button, then click on the Sampling button and change the Maximum Sample Size to 100000000 (you can do this by just backspacing over the rightmost zero in the Maximum Sample Size box), click the OK button in the Data Sampling window, then click the OK button in the Classification window, and finally click the OK button in the Layer Properties window.

the model cell intersected by the well. Let's refer to the result of this calculation with the term, T_i, where i represents a given model layer. Note that if the well is open to the entire thickness of a particular model layer, i, then T_i will be equal to the transmissivity of the model layer in model cell penetrated by that well.

2. Compute the sum of all of the $T_i$values calculated in the previous step: $\sum_{i=1}^{4} T_i$
3. Compute the estimated withdrawal rate using the following formula:

$$W_i = W * \left(\sum_{i=1}^{4} T_i\right),$$

Where W represents the total withdrawal from the multilayer well, and W_i represents the rate of withdrawal from the well in model layer i.

Appendix B: Geographic Projection Information for horizontal coordinates in file, sim_cup_input.xlsx and sim_cup_input.csv

The values for the horizontal coordinates in the file, sim_cup_input.csv, are expected to be in one of two projection systems: (1) Florida State Plane North or (2) UTM Zone 17 North. At the SRWMD, the default projection system is the 'Florida State Plane North' projection system, so the horizontal (x,y) coordinates described in paragraph 3.c of section, for new wells should be specified for this Florida State Plane North projection system. This projection system is itself specified as follows:

```
Projected Coordinate System:
     NAD_1983_HARN_StatePlane_Florida_North_FIPS_0903_Feet

Projection:       Lambert_Conformal_Conic

False_Easting:    1968500.00000000

False_Northing:   0.00000000

Central_Meridian:      -84.50000000

Standard_Parallel_1:   29.58333333

Standard_Parallel_2:   30.75000000

Latitude_Of_Origin:    29.00000000

Linear Unit:      Foot_US


Geographic Coordinate System:    GCS_North_American_1983_HARN

Datum:      D_North_American_1983_HARN

Prime Meridian:   Greenwich

Angular Unit:     Degree
```

The above information can also be represented as an ArcGIS projection file (prj) format:

```
PROJCS["NAD_1983_HARN_StatePlane_Florida_North_FIPS_0903_Feet",GEOGCS[
"GCS_North_American_1983_HARN",DATUM["D_North_American_1983_HARN",SPHE
ROID["GRS_1980",6378137.0,298.257222101]],PRIMEM["Greenwich",0.0],UNIT
["Degree",0.0174532925199433]],PROJECTION["Lambert_Conformal_Conic"],P
ARAMETER["False_Easting",1968500.0],PARAMETER["False_Northing",0.0],PA
RAMETER["Central_Meridian",-
84.5],PARAMETER["Standard_Parallel_1",29.58333333333333],PARAMETER["St
andard_Parallel_2",30.75],PARAMETER["Latitude_Of_Origin",29.0],UNIT["F
oot_US",0.3048006096012192]]
```

At the SJRWMD, the default projection system is the 'UTM Zone 17 North' projection system, so the horizontal (x,y) coordinates described in paragraph 3.c of section, can be specified in this UTM Zone 17 North projection system, provided that the sim_cup_sjr.bat file is used to carry out the model preprocessing, execution, and postprocessing. This UTM Zone 17 North projection system is specified as follows:

```
Projected Coordinate System:     NAD_1983_HARN_UTM_Zone_17N

Projection:      Transverse_Mercator

False_Easting:   500000.00000000

False_Northing:  0.00000000

Central_Meridian:     -81.00000000

Scale_Factor:    0.99960000

Latitude_Of_Origin:   0.00000000

Linear Unit:     Meter


Geographic Coordinate System:    GCS_North_American_1983_HARN

Datum:     D_North_American_1983_HARN

Prime Meridian:  Greenwich

Angular Unit:    Degree
```

The above information can also be represented as an ArcGIS projection file (prj) format:

```
PROJCS["NAD_1983_HARN_UTM_Zone_17N",GEOGCS["GCS_North_American_1983_HA
RN",DATUM["D_North_American_1983_HARN",SPHEROID["GRS_1980",6378137.0,2
98.257222101]],PRIMEM["Greenwich",0.0],UNIT["Degree",0.017453292519943
3]],PROJECTION["Transverse_Mercator"],PARAMETER["False_Easting",500000
.0],PARAMETER["False_Northing",0.0],PARAMETER["Central_Meridian",-
81.0],PARAMETER["Scale_Factor",0.9996],PARAMETER["Latitude_Of_Origin",
0.0],UNIT["Meter",1.0]]
```