

Metro Model 3 User Manual and Tutorial Guide

March, 2014

Table of Contents

Introduction.....	2
Tutorial 1: Navigating Metro Model 3 in Groundwater Vistas.....	3
1.1 Introduction.....	3
1.2 Model Units and Grid.....	3
1.3 First Opening Groundwater Vistas.....	3
1.4 Importing and Creating Reference Maps	3
1.5 Importing Model Results.....	4
1.6 Exporting Model Results.....	5
1.7 Wells	5
1.8 Hydraulic Conductivity and Hydrostratigraphy	7
1.9 Infiltration	8
1.10 Model Mass Balance Summary Information	8
1.11 Advanced MODFLOW Settings and Packages	9
1.11.1.1 MODFLOW-NWT Settings	9
1.11.1.2 Quasi-3D Confining Beds.....	9
1.11.1.3 Unsaturated Zone Flow (UZF) Package	9
1.11.2 Transient Simulations	10
1.11.2.1 Stress Periods.....	10
1.11.2.2 Storage	10
1.11.2.3 Transient infiltration	10
1.11.2.4 Transient Pumping Data	10
Tutorial 2: Evaluate Impacts from a New Well	11
2.1 Introduction.....	11
2.2 Getting Started	11
2.3 Open and run current version of model.....	11
2.4 Save new model for comparison	12
2.5 Add new well	12
2.5.1 Importing Well Information (Optional).....	14
2.6 Run Updated Model	15
2.7 Evaluate Drawdown.....	15
2.8 Evaluate Changes in Baseflow	18
Tutorial 3: Model Refinement and Capture Zone Analysis	20

3.1 Introduction	20
3.2 Getting Started	20
3.3 Refine Model Grid	20
3.4 Add Local Hydrogeologic Data	21
3.4.1 Manually Adjust Parameters.....	21
3.4.2 Import New/Updated Data from GIS.....	22
3.5 Evaluate Drawdown and Well Interference	25
3.6 Assess Capture Zone of New Well using Particle Tracking	26
Tutorial 4: Implementing Lake Package	29
4.1 Introduction	29
4.2 Getting Started	29
4.3 Implementing the Lake Package	29
4.4 Evaluating Lake Levels.....	31
Tutorial 5: Modifying SWB Model	32
5.1 Introduction	32
5.2 Getting Started	32
5.3 Modify Climate Data.....	33
5.3.1 Generate Baseline Synthetic Climate Data Set.....	34
5.3.2 Convert Baseline Synthetic Data Set to SWB Climate Time Series Format.....	35
5.3.3 Generate Climate Change Synthetic Data Set	36
5.3.4 Convert Climate Change Synthetic Data Set to SWB Climate Time Series Format.....	37
5.4 Modify Land Use/Land Cover	38
5.5 Modify SWB Control File.....	44
5.5.1 Setting Climate Data in Control File	44
5.5.2 Setting New Land Use Grid.....	44
5.6 Running SWB	45
5.7 Comparing Results.....	46
5.8 Using Gridded Climate Data.....	51
Tutorial 6: Importing Files into Other Graphical User Interfaces.....	52
6.1 Introduction.....	52
6.2 Groundwater Modeling System (GMS)	52
6.3 Model Muse	53
6.4 Visual MODFLOW.....	53
References and Additional Resources.....	54

Introduction

Metro Model 3 is a nine-layer regional groundwater-flow model that simulates groundwater flow in the eleven-county Twin Cities area. The tutorials in this user manual present the most commonly used applications of Metro Model 3. The methods used are not the “answer” to any particular problem but are presented as a guide to the general process and to provide tips for items unique to the regional model. It is assumed that users of the Metro Model 3 have a working knowledge of groundwater-flow models and in particular, MODFLOW; therefore, the tutorials do not describe in detail *how* processes/model packages work.

All tutorials are set up using the graphical user interface (GUI)/modeling environment “Groundwater Vistas.” Although this GUI was selected for the development of the model, its use does not imply an endorsement of this product. Exercise and answer files are provided for each tutorial in individual directories included with this document. The files were created using Groundwater Vistas version 6.58 build 5 or earlier. Users may need to update their software to use the provided files. Groundwater Vistas may be purchased from Environmental Simulations Incorporated (ESI) at www.groundwatermodels.com.

This user manual contains six tutorials. Tutorial 1 teaches users how to navigate Metro Model 3 in Groundwater Vistas, showing how data are stored within Groundwater Vistas and demonstrating how to view and export model results. Tutorials 2-4 build on each other, adding more detail in each tutorial. Tutorial 2 demonstrates adding a well to the model and evaluating drawdown and change in river flux. Tutorial 3 assesses the impact from the same well by implementing a refined model grid and adjusted aquifer properties. This tutorial also demonstrates the use of particle tracking to determine the capture zone of a well. Tutorial 4 illustrates use of the Lake Package to estimate change in lake level associated with pumping. Tutorial 5 shows the user how to estimate infiltration using the SWB model within Metro Model 3, and Tutorial 6 teaches the user how to import MODFLOW files for Metro Model 3 into other user interfaces. It is recommended that the user progress through Tutorials 1-4 in order; however, Groundwater Vistas files are provided for users who would like to jump ahead.

A list of references and additional resources is provided in the References and Additional Resources section of this user manual. Full details of Metro Model 3 construction and calibration may be found in the main model report (Metropolitan Council, 2014).

Tutorial 1: Navigating Metro Model 3 in Groundwater Vistas

1.1 Introduction

Tutorial 1 provides basic background information regarding Metro Model 3 and how data for the model are stored within Groundwater Vistas. This tutorial is designed to be both a reference for advanced users and guide on basic functionality for novice users. Novice users are encouraged to review the Groundwater Vistas manual and associated tutorials for further explanation of how Groundwater Vistas works. For details regarding the conceptual model, model construction, and calibration see Metropolitan Council (2014).

1.2 Model Units and Grid

The Metro Model 3 uses length units of meters and time units of days. MODFLOW cannot mix units. However, Groundwater Vistas can convert some units on the fly for convenience. In general, a user should not attempt to work in multiple unit systems (e.g., SI units and gpm) because this can often result in hard-to-discover errors that propagate through the entire simulation process. The model uses a regular grid with dimensions of 500 meters x 500 meters. The coordinate system used is UTM NAD1983 Zone 15N. The origin of the model grid (lower left) is located at easting 369,875 meters and northing 4,883,875 meters. When importing data into Groundwater Vistas (wells, boundaries, etc.) the coordinate system should be carefully considered. While Groundwater Vistas can convert between some coordinate systems, it is best to convert all GIS data to UTM NAD 1983 Zone 15N prior to importing. Again, using consistent coordinate systems and units makes for good model “hygiene”.

To turn the grid display on or off select **Plot | What to Display** from the main menu then check the box labeled **Display Finite-Difference Grid**

1.3 First Opening Groundwater Vistas

When first opening a Groundwater Vistas file supplied as part of this tutorial or one of the main Metro Model 3 files (steady-state or transient) it will likely give a warning that the working directory does not exist. This is expected.

1. Click “Yes”
2. Select any directory where you wish to have model files saved.

Groundwater Vistas may also indicate that certain map files cannot be opened. Again, this is expected.

1. Click “Yes”
2. Two Groundwater Vistas map files are provided: 1) *counties.map* and 2) *roads.map*. These files are located in the `.\Groundwater_Vistas_Map_Files` directory that is packaged with the distributed model files. Select the `.map` file it is asking for
- 3.

1.4 Importing and Creating Reference Maps

Groundwater Vistas is able to display reference maps to help orientate the user. However, these maps are not stored within Groundwater Vistas and must be read in separately. Two Groundwater Vistas map files are provided: 1) *counties.map* and 2) *roads.map*. These files are located in the `.\Groundwater_Vistas_Map_Files` directory that is packaged with the distributed model files.

To import the map files into Groundwater Vistas:

1. Select **File | Map | GW Vistas** on the main menu
2. Navigate the `.\Groundwater_Vistas_Map_Files` directory
3. Select one of the map files provided and select **Open**.

Additional maps can be created using GIS shapefiles. These shapefiles should be in coordinate system NAD 1983 Zone 15 N. To create a new map in Groundwater Vistas:

1. Select **File | Map | Shapefile** on the main menu.
2. Navigate to a shapefile to be converted and select **Open**.
3. Groundwater Vistas will then ask to save the `.map` file. Choose any convenient file location.
4. An option to select a color will be presented and then the newly created map will be added to the interface.

1.5 Importing Model Results

It is often desirable to import model results without running the model. For instance, you may wish to view groundwater contours or fluxes and have no specific reason to modify the model. To accommodate this type of use the model output files for the Metro Model 3 as distributed are provided. To import model results:

1. Select **Plot | Import Model Results**.
2. Click the **Browse** button for the head file and navigate to the directory where model files are stored and select the `*.hds` file.
3. Select **OK** to read in the heads.

Cell-by-Cell Flow files can also be imported in a similar manner. The Cell-by-cell flow file contains all information regarding fluxes between cells and boundaries within the model and are required to compute mass balance information. Cell-by-cell flow files generated by running the model within Groundwater Vistas have the file extension `*.cbb`

To display contours of hydraulic head:

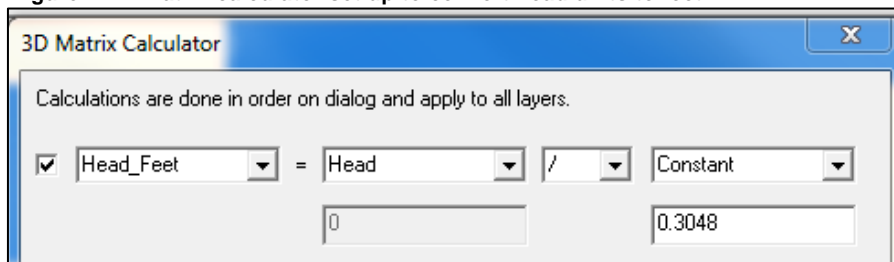
1. Select **Plot | What to Display**. Make sure the box for **Display Contours of** is checked and Head is selected from the drop down box.
2. The contour interval can be adjusted by selecting **Plot | Contour | Parameters (plan)** and selecting preferred options in the Contour Information dialog box

By default all contours are in units of meters. To display contours of feet rather than meters¹:

1. Select **Plot | Matrix Calculator | Compute 3D**. Set up a calculation as shown below. Variable names can be modified from Variable 1, Variable 2, etc. using **Plot | Matrix Calculator | Variable Names**.

¹ Groundwater Vistas allows users to view results in units other than the model's native system of meters, but the user must use the **Matrix Calculator** to perform the unit conversions. These unit changes are for viewing results in different units but do not affect any model input values.

Figure 1.4-1 Matrix calculator set up to convert head units to feet



2. Once the calculation is set up, select **Plot | What to Display**. In the drop down box next to **Display Contours of** select *Head_Feet* (or any other named variable set up in the Compute 3D dialog box). Contours will now be displayed in units of feet. Model results may need to be reimported for the conversion to be calculated.

1.6 Exporting Model Results

Model results and data can be exported from Groundwater Vistas in many ways-- too numerous for this document to address completely. We will cover how to export shapefiles of model contours for use in GIS programs, as this is a common post-processing task.

1. Import model results as described in Section 1.5
2. Select **Plot | What to Display** and make sure **Display Contours of Head** is selected.
3. Select **File | Export** enter a file name and select **Shapefile** for the datatype.
4. In the Export Model Results to an ArcView Shapefile dialog window, select **Result Type** as contours and select **OK**. The coordinate system used is UTM NAD1983 Zone 15N. However, this information is not retained in the GIS files exported from Groundwater Vistas. The user should note this and set the appropriate spatial reference when viewing data in their GIS software.

Note: Contours are exported as displayed and for the current layer. The contour interval can be adjusted by selecting **Plot | Contours | Parameters (Plan)**. The contour interval attribute value is also exported in the associated .dbf file.

1.7 Wells

All wells within Groundwater Vistas are implemented using “analytic elements”. Analytic elements in Groundwater Vistas are grid independent boundary conditions. Analytic element wells are used to map wells to exact coordinates so that if the grid is modified, wells will be assigned to the appropriate model cell.



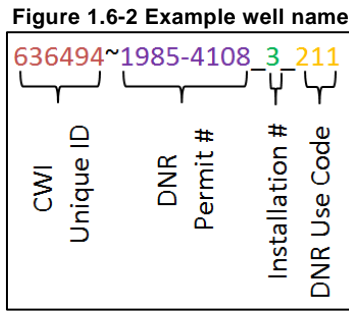
Properties for individual wells can be viewed/edited by first selecting the **edit analytic elements** button  and then double clicking on a well. You may need to zoom into a well to be able to click on it. Wells are symbolized with the  symbol.

Figure 1.6-1 Well Information dialog box

Even though all wells are stored internally in Groundwater Vistas with analytic elements, two separate well packages are created for MODFLOW: the multi-node well package (MNW) for wells that penetrate multiple layers and the standard well package (WEL) for wells that are only open to a single layer. In the well information dialog box if the top layer and bottom layer for a well are not equal then a check mark should be present for “Use as Fracture Well or Multi-Node Well”. Checking this box indicates that this well should be included in the MNW Package. If the box is not checked it will be included in the standard WEL Package.

All **single aquifer wells** are identified by a concatenation of the County Well Index (CWI) unique ID, the DNR permit number, the DNR installation ID, and the DNR use code. This naming scheme was developed to allow for the use of the commonly used CWI unique IDs while still accommodating those wells that do not have a CWI unique ID listed in the SWUDS database. The use of the DNR permit number, installation ID, and use code was required to achieve a complete set of unique names (no duplicates). Due to a limitation on the width of well names in the MNW2 package, well identifications for **multi-aquifer wells** are defined with shorter names. A lookup table relating the well names for multi-aquifer wells and the identification system described above is provided in Appendix D of the main model documentation (Christianson and Wuolo, 2014).


The following example demonstrates the naming system.




The CWI unique ID and the DNR permit number are always separated by a “~” sign. If the well does not have a CWI unique ID then the well name starts with a “~” sign. The DNR use code is always the last three integers. In the example above, the use code of “211” indicates that the well is used for municipal supply. The installation code of “3” indicates that it is the third well included in permit number 1985-4108. The CWI unique ID for this well is 636494, which can be used to look up additional well-construction data and drilling records in the CWI database.

1.8 Hydraulic Conductivity and Hydrostratigraphy

Hydraulic conductivity values are stored in **matrix format** within Groundwater Vistas, meaning values are assigned to individual model cells and are not based on discrete zones. Defining hydraulic conductivity in matrix format is necessary to allow for calibration using pilot points. However, this approach makes it more difficult to visualize the extent of individual hydrostratigraphic units and to manually make adjustments to hydraulic conductivity. To overcome these limitations the hydrostratigraphy property is used in conjunction with the hydraulic conductivity property when viewing model results and manipulating hydraulic conductivity values.

To view hydraulic conductivity values, the hydraulic conductivity property needs to be active. This can be done by selecting **Props | Hydraulic Conductivity** from the main menu or by clicking the edit **Property Zones Button**  and then selecting **Hydraulic Conductivity** in the dropdown box. When the hydraulic conductivity property is active, values for individual cells will display along the left side of the status bar at the bottom of the screen.

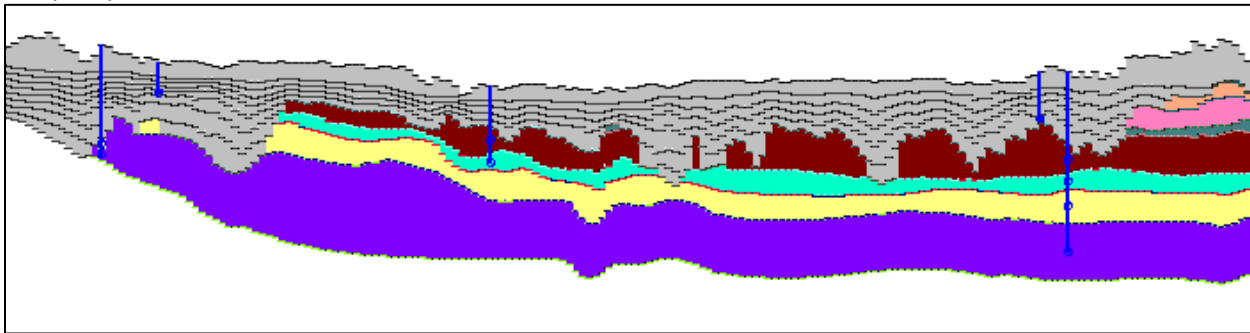
To view the extent of individual hydrostratigraphic units, the hydrostratigraphy property should be active. Select **Props | Hydrostratigraphy** from the main menu, or click on the **Property Zone Button**  and then select **Hydrostratigraphy** from the drop down menu. Each major hydrostratigraphic unit is represented by a hydrostratigraphy zone number and each zone number is displayed as a separate color. Zone values are shown along the left side of the status bar at the bottom of the screen.

Zone numbers are as follows:

Table 1.7-1 Hydrostratigraphy property zone numbers

<i>Zone Number</i>	<i>Hydrostratigraphic Unit</i>
1	Quaternary Sediments (undifferentiated)
2	St. Peter Sandstone
3	Prairie du Chien Group
4	Jordan Sandstone
5	St. Lawrence Formation
6	Tunnel City Group
7	Wonewoc Sandstone
8	Eau Claire Formation
9	Mt. Simon Sandstone
10	Cretaceous and unnamed Paleozoic

Figure 1.7-1 Cross section view with hydrostratigraphy property active. Each color indicates a separate hydrostratigraphic unit (zone)



1.9 Infiltration

Infiltration for the UZF Package is stored in matrix format similar to hydraulic conductivity. Infiltration values can be viewed and edited by selecting **Props | Recharge**. These values represent infiltration rates used by the UZF Package which simulates flux of water through the unsaturated zone and provides recharge to the saturated groundwater flow system. Units for infiltration are m/day. MODFLOW will internally convert infiltration to a volume based on the area of each model cell.

1.10 Model Mass Balance Summary Information

To view mass balance information the cell-by-cell flow file must first be imported.

- 1.** Select **Plot | Import Model Results**.
- 2.** Click the **Browse** button for the Cell-by-Cell Flow file, navigate to the directory where model files are stored, and select the *.cbb file.
- 3.** Make sure the box to import the cell-by-cell flow file is checked
- 4.** Select **OK** to read in the cell-by-cell flow.

Once the cell-by-cell flow file has been imported, mass balance information can be viewed.

- 5.** Select **Plot | Mass Balance | Model Summary**. A window presenting the total inflows and outflows for each boundary condition in the model is presented.

6. In the MODFLOW Mass Balance dialog window select the **Graph** button in the upper right corner. After clicking the graph button the data will be displayed as a bar chart.

1.11 Advanced MODFLOW Settings and Packages

Metro Model 3 uses some advanced MODFLOW options that are not very intuitive to use with Groundwater Vistas. The Groundwater Vistas file distributed with Metro Model 3 already has all settings for these advanced options implemented. These advanced settings are described below so that the user is aware of them and use them when troubleshooting any changes they make.

1.11.1 MODFLOW-NWT Settings

Metro Model 3 uses MODFLOW-NWT to allow for sophisticated handling of dry cells. There are three important items to consider when using MODFLOW-NWT.

- 1.) MODFLOW-NWT requires the use of the Upstream Weighting Package (UPW) and the Newton Solver (NWT). The UPW and NWT packages are turned on by checking the box in the MODFLOW2005 Packages dialog box. Select **Model | MODFLOW2005 | Packages** and check the box **Use Upstream Weighting Package**.
- 2.) Solver settings for the NWT solver are accessed by selecting **Model | MODFLOW2005 | Options** and clicking on the **NWT General** tab and **NWT Methods** tab
- 3.) Resaturation must be set as inactive by selecting **Model | MODFLOW | Package Options** then selecting the **Resaturation** tab, making sure that the box for *Resaturation Capability is Active* is unchecked.

1.11.1.1 Quasi-3D Confining Beds

Quasi-3D confining beds are used to simulate confining characteristics of the base of the St. Peter Sandstone, Prairie du Chien Group, and Tunnel City Group, where present. Because these units may be represented in different model layers due to faulting, quasi-3D layers are between each model layer. Each quasi-3D confining layer is set at a constant thickness of 0.1 meters. This thickness is set for convenience in computation of conductance and does not represent the actual thickness of confining beds. Where confining units are present, the vertical hydraulic conductivity of the quasi-3D layer is set low to reduce the conductance between layers. Where confining units are not present, the vertical hydraulic conductivity is set high so that the quasi-3D layer has no significant effect on the conductance between layers.

Quasi-3D layers are set active within Groundwater Vistas by selecting **Model | MODFLOW | Package Options** and setting **Leakance Zones Represent Kz of Aquitard** on the **BCF-LPF** tab

The vertical hydraulic conductivity of quasi-3D layers is defined by the **leakance property**. Select **Props | Leakance**. For each layer, in areas where a quasi-3D unit is present a low, Kz (shown as Leakance on the status bar) will be present. Areas where no quasi-3D confining layer is present will have a vertical hydraulic conductivity value of 1.0E+04.

1.11.1.2 Unsaturated Zone Flow (UZF) Package

The UZF package is used in Metro Model 3 to account for the time lag (for transient simulations) between infiltration occurring below the root zone and recharge reaching the water table. This is a pressure or moisture-content time lag and not a water-particle time-of-travel. The UZF package also allows for infiltration to be rejected if the water table is at the ground surface (e.g. large wetland complexes).

The UZF Package is set active by selecting **Model | MODFLOW2005 | Packages** and checking the box to create the UZF1 package. General UZF options are located at **Model | MODFLOW2005 | Options** under the **UZF** tab.

The vertical hydraulic conductivity of the unsaturated zone and Brooks-Corey epsilon values for the UZF package are stored in the **Props | Unsaturated Zone** properties. Saturated water content of the unsaturated zone for the UZF package is set from the porosity property which is located in **Props | Storage Property**.

Infiltration for the UZF package is set from the recharge property located in **Props | Recharge**.

1.11.2 *Transient Simulations*

All tutorials in this document are designed to be run using the steady-state version of Metro Model 3. This is to alleviate potential frustration resulting from extremely long transient model run-times. Details regarding transient simulations are provided here for reference.

The Groundwater Vistas file `.\MM3_Transient_1995_2011.gvw` is set up to run a transient simulation from 1995 to 2011 as described in Metropolitan Council(2014). Transient data are not available in the file `.\MM3_Steady_State.gvw`.

1.11.2.1 *Stress Periods*

The transient version of Metro Model 3 provided has 205 stress periods. The first stress period is a steady-state stress period representing 1990-1995. Stress periods 2-205 are monthly stress periods starting January 1995 and ending December, 2011. Each transient stress period has four time steps. Stress period and time step length can be viewed by selecting **Model | MODFLOW | Stress Period Setup**. A **time step multiple of 1.0** is recommended for all transient stress periods to avoid potential issues with the timing of recharge as calculated using the UZF Package.

1.11.2.2 *Storage*

Aquifer storage properties can be viewed/edited by selecting **Props | Storage/Porosity**. Like hydraulic conductivity, storage properties are stored in matrix format. See Section 1.8 for details how to use the hydrostratigraphy property to help navigate and visualize hydrostratigraphic units.

1.11.2.3 *Transient infiltration*

Infiltration for the UZF Package is stored in matrix format similar to hydraulic conductivity. Infiltration values can be viewed and edited by selecting **Props | Recharge**. To view changes in infiltration for each stress period, adjust the stress-period number in the navigation window on the left side of the screen.

1.11.2.4 *Transient Pumping Data*

As described in Section 1.7, properties for individual wells can be viewed/edited by first selecting the **edit analytic elements** button and then double clicking on a well. In the Well Information dialog window pumping rates for each stress period can be viewed/edited by clicking on the **Transient Data** button under **Well Options**.

Tutorial 2: Evaluate Impacts from a New Well

2.1 Introduction

Tutorial 2 is designed to familiarize the user with basic modifications to Metro Model 3 and presents methods to help analyze results of different model predictions. In this tutorial a new well will be added to the Prairie du Chien – Jordan aquifer and pumped at 500 gpm. Changes in baseflow to a nearby stream and levels of drawdown will be analyzed. Typically for this type of problem, the model grid will need to be refined, especially if the user is interested in predictions very near the proposed well. However, for this tutorial it is assumed that grid refinement is not necessary. Methods to refine the model grid are presented in Tutorial 3.

2.2 Getting Started

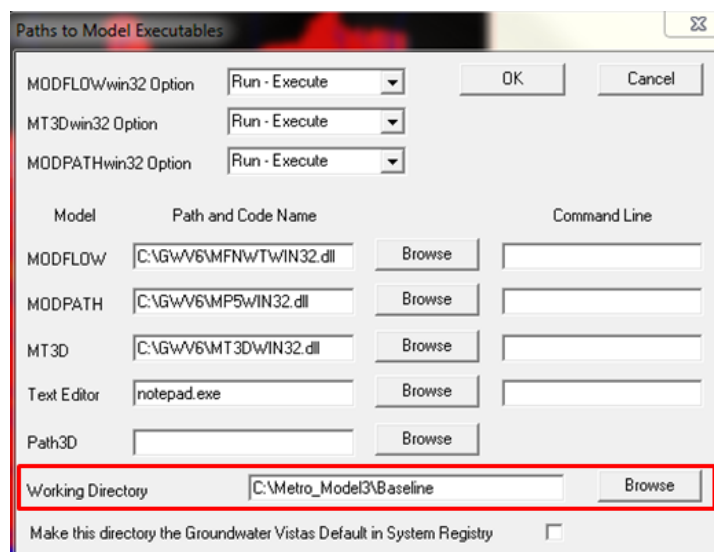
Two Groundwater Vistas files are provided at *MM3_User_Manual\Tutorials\Tutorial2*. The file *MM3_Steady_State_Baseline* is the calibrated steady-state version of Metro Model 3 and is the starting point for this tutorial. The file *MM3_Steady_State_New_Well.gvw* is a model derived from completion of this tutorial and is provided for reference.

2.3 Open and run current version of model

In order to analyze the effects of adding a new well, a baseline simulation must be run and saved to use for comparison. We will use the calibrated steady-state version of Metro Model 3 as the baseline simulation.

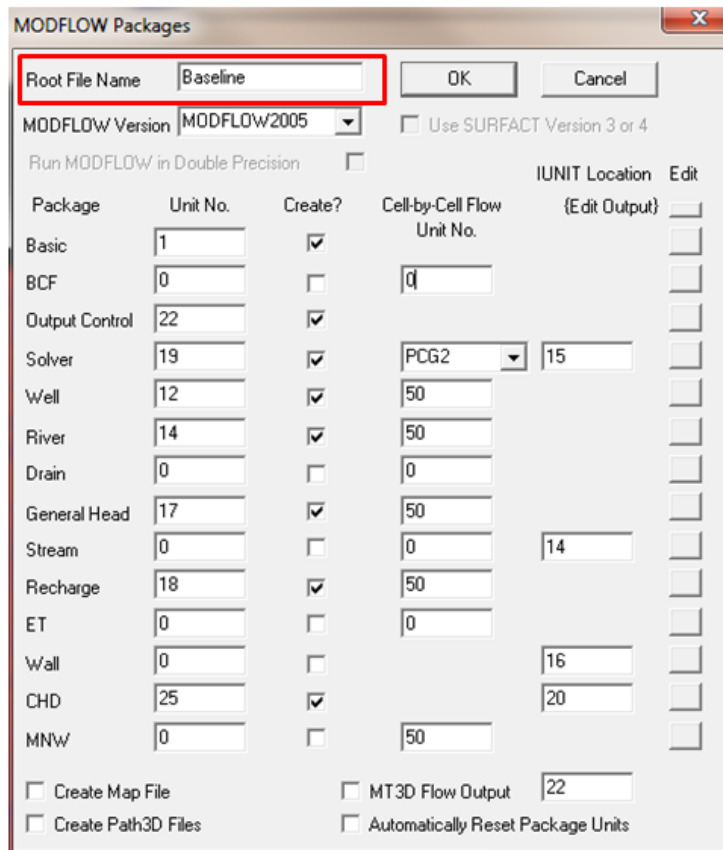
1. Open the Groundwater Vistas file of the calibrated steady-state version of Metro Model 3 (*MM3_Steady_State_Baseline.gvw*) and set the working directory by selecting **Model | Path to Models** and enter any convenient directory in the Working Directory dialog box.


Figure 2.3-1 Working Directory dialog box



2. Set the Root File Name by selecting **Model | MODFLOW | Packages** and entering any convenient name in the Root File Name dialog box.

Figure 2.3-2 Root File Name dialog box



3. Create datasets and run the baseline model by clicking the **Recalculate** or **Run Model** button 
4. After the model has completed running the Import Model Results dialog window should open. Make sure the path to the Head File is correct and select OK to import model results.

2.4 Save new model for comparison

Now that a baseline simulation has been created and run, we will create a new model with an additional well to use for comparison. While a new Groundwater Vistas file is not necessarily required, we will create one so that changes can be saved along the way.

1. In Groundwater Vistas click **File | Save As** and save a new Groundwater Vistas file.
2. Follow **Step 1 and Step 2 from Section 2.3** above and reset the working directory and root file name to distinguish the new model developed for this tutorial from other models.

2.5 Add new well

As described in Section 1.7, wells are stored within Groundwater Vistas as analytic elements. Analytic element wells are used to map wells to exact coordinates so that if the grid is modified, wells will be assigned to the correct model cell. We will add a well open to the Prairie du Chien – Jordan aquifer (model Layers 3 and 4) in the model cell located at Row 210 Column 200 and designate a steady-state pumping rate of 2,725 m³/day (500 gpm).

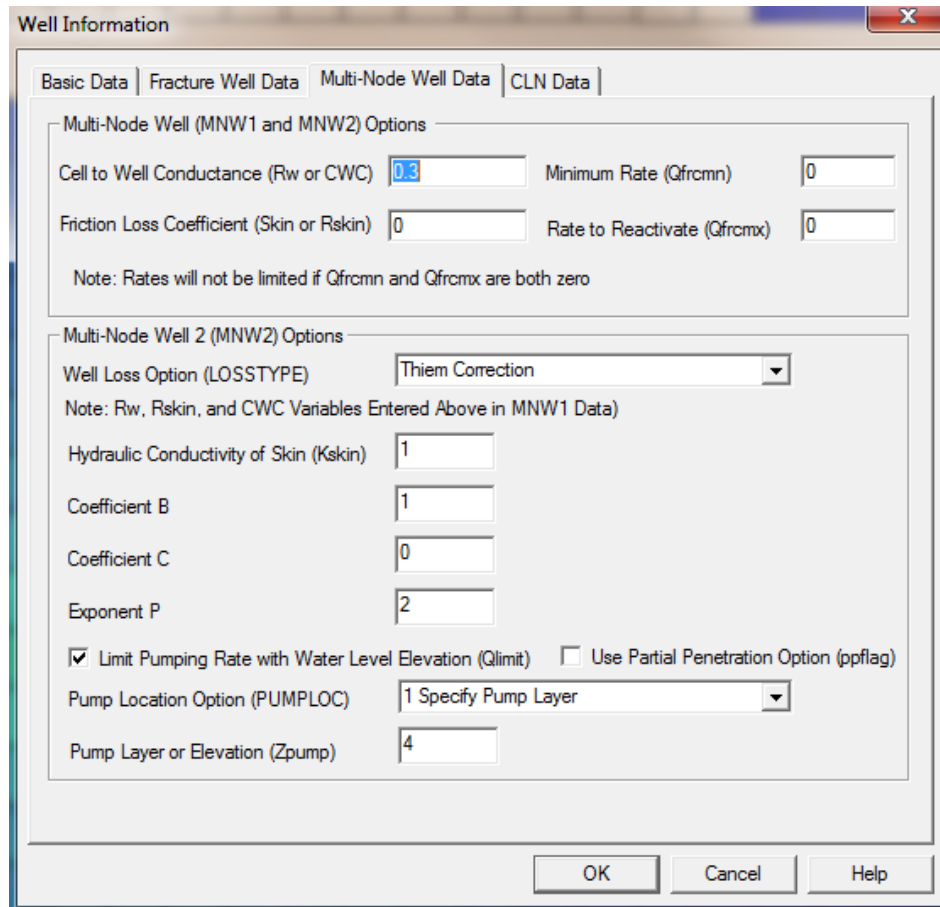
1. Select **AE | Well** and then click in Row 210 Column 200 to add a new well. After clicking the Well Information dialog window will pop up.
2. In the Well Information dialog window under the **Basic Data** tab, enter in values as shown in Figure 2.5-1. If desired, a well name may be assigned to the new well. However, well names with spaces or well names greater than 20 characters are not acceptable for multi-node wells. It is important to remember sign conventions associated with MODFLOW; water leaving the simulation (i.e. extraction) is designated with a negative value, water entering the simulation is designated as positive. We are simulating an extraction of water so the pumping rate specified must be negative.

Figure 2.5-1 Basic Data for new well

The screenshot shows the 'Well Information' dialog box with the 'Basic Data' tab selected. The 'Spatial Parameters' section includes X: 469600, Y: 4984100, Top Layer of Screen: 3, and Bottom Layer of Screen: 4. The 'Well Options' section includes Steady-state Pumping Rate: -2725, Concentration: 0, Component: 1, and Well Type: Standard. The 'Well-specific settings' section includes Use as Fracture Well (FWL4 or 5) or Multi-Node Well (MNW) or CLN (MF-USG) checked, Pumping Level for FWL4 or MNW: 145, and Reach Number: 9999. The 'Use with FWL5' checkbox is unchecked. The 'Well Name' field contains 'NewWell1'.

3. In the Well Information dialog window under the **Multi-Node Well Data** tab, enter in values as shown in Figure 2.5-2 and then select **OK**. For additional information on well parameters associated with multi-node wells see the Groundwater Vistas help system and Konikow and others (2009).

Figure 2.5-2 Multi-Node Well Data tab



2.5.1 Importing Well Information (Optional)

It is often desirable to import well locations and pumping information from a separate file rather than adding them individually within the Groundwater Vistas GUI. While direct importation of GIS shapefiles is possible within Groundwater Vistas, all options for multi-node wells (MNW) are not supported. If well data are in a shapefile, the attribute table (including X and Y coordinates) should be exported to a text file (typically .csv) and then imported into Groundwater Vistas. The procedure below describes how to import from a text file but is not necessary for this tutorial if the “New Well” was already added as outlined above.

An example shapefile for a well is provide at:

.\MM3_User_Manual\Tutorials\Tutorial2\Support_Files\New_Well.shp

The attributes of this shapefile in .csv format is provided at:

.\MM3_User_Manual\Tutorials\Tutorial2\Support_Files\New_Well.csv

The .csv file can be imported into Groundwater Vistas as follows.

1. Select **AE / Import / Well Text File**. Navigate to the file *New_Well.csv*
2. In the Options for Importing Wells dialog window select column numbers for appropriate fields as shown in Figure 2.5-3

Figure 2.5-3 Multi-Node Well Data tab

Options for Importing Wells

Wells are in Site Coordinates

File Contains Transient Wells

Set as a Fracture Well, MNW, or CLN Well

Set as a Monitoring Well

Well Type: Standard

Number of Lines to Skip: 1

Parameter	Column in File	Parameter	Column in File	Parameter	Column in File
Name	2	No. Trans. Data Pts.	0	Conductance (Rw)	6
X Coordinate	13	Column	0	Friction Loss (Skin)	0
Y Coordinate	14	Row	0	Min. Rate (Qfrcmn)	0
Screen Top	0	Top Layer	5	Q Reactivate (Qfrcmx)	0
Screen Bottom	0	Bottom Layer	4	Is MNW or FWL?	12
Flow Rate	3	Concentration	0	Qlimit (MNW2)	0
Type	0	Screen radius	0	Kskin (MNW2)	0
Pumping Level	11	Casing radius	0	Loss Type (MNW2)	7
Reach No.	0	Storage (fwl4)	0	Coef. B (MNW2)	0
				Coef. C (MNW2)	0
				Exponent P (MNW2)	0
				PUMPLOC (MNW2)	9
				Zpump (MNW2)	10

All Data on One Line with Transient Data at End of Line

Multiply All Well Rates by -1.0

Allocate Pumping Rates Based on Screen Elevations

Include Storage Effects


Buttons: OK, Cancel, View File, Browse...

Groundwater Vistas can sometimes have trouble importing very large well datasets from a text file. Some tips to try if having problems importing include:

- Only include fields needed for importing and defining well properties. Delete all extraneous fields that are not imported.
- Include a blank line at the end of the text file. Sometimes if a single well is not imported adding a blank line typically fixes this issue.
- Try a file with a different delimiter. For example, switch from commas to tabs or vice versa.

2.6 Run Updated Model

After adding the new well, the model must be run to generate output to use in comparing to the baseline simulation created in Section 2.3

1. Select **File | Save** to save any changes made. This is not required but it is good practice.
2. Create datasets and run the model by clicking the **Recalculate** or **Run Model** button 

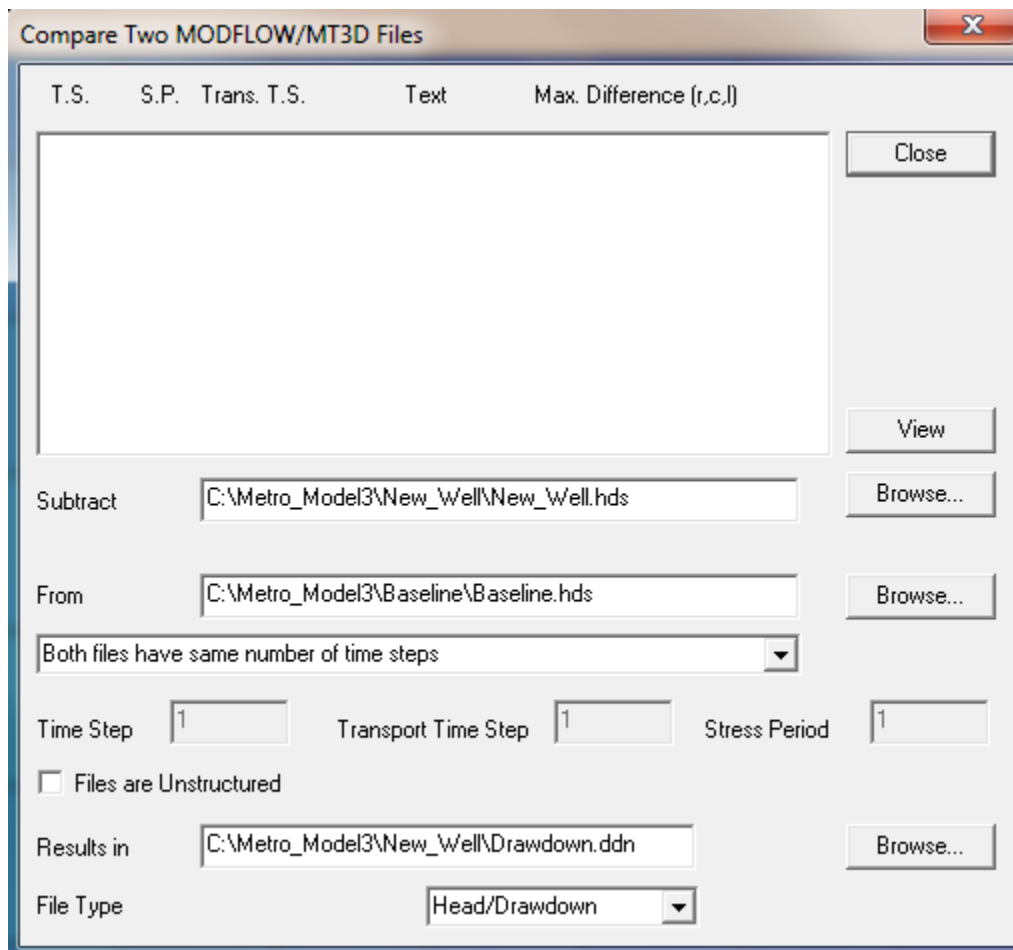
2.7 Evaluate Drawdown

Now that we have two models, a baseline simulation and a simulation with a new well, we can compare both heads and flow to assess the effects of pumping the new well.

First we will evaluate changes in head or drawdown. There are several ways to do this. Traditionally, MODFLOW is used to output a drawdown file. However, this requires that the starting heads be set correctly. Experience has shown that this method often leads to errors in model setup, particularly for novice modelers. Instead, we will use utilities built into Groundwater Vistas to compare the binary head files created for each of the two simulations (baseline and new well).

1. Select **Plot | File Operations | Compare Two Files**.
2. Enter information into the Compare Two MODFLOW/MT3D Files dialog window, as shown in Figure 2.7-1. We are subtracting the new well head file (*New_Well.hds*) from the baseline head file (*Baseline.hds*) to compute drawdown. Results will be saved into a new file called *Drawdown.ddn*.

Figure 2.7-1 Compare files dialog window for computing drawdown

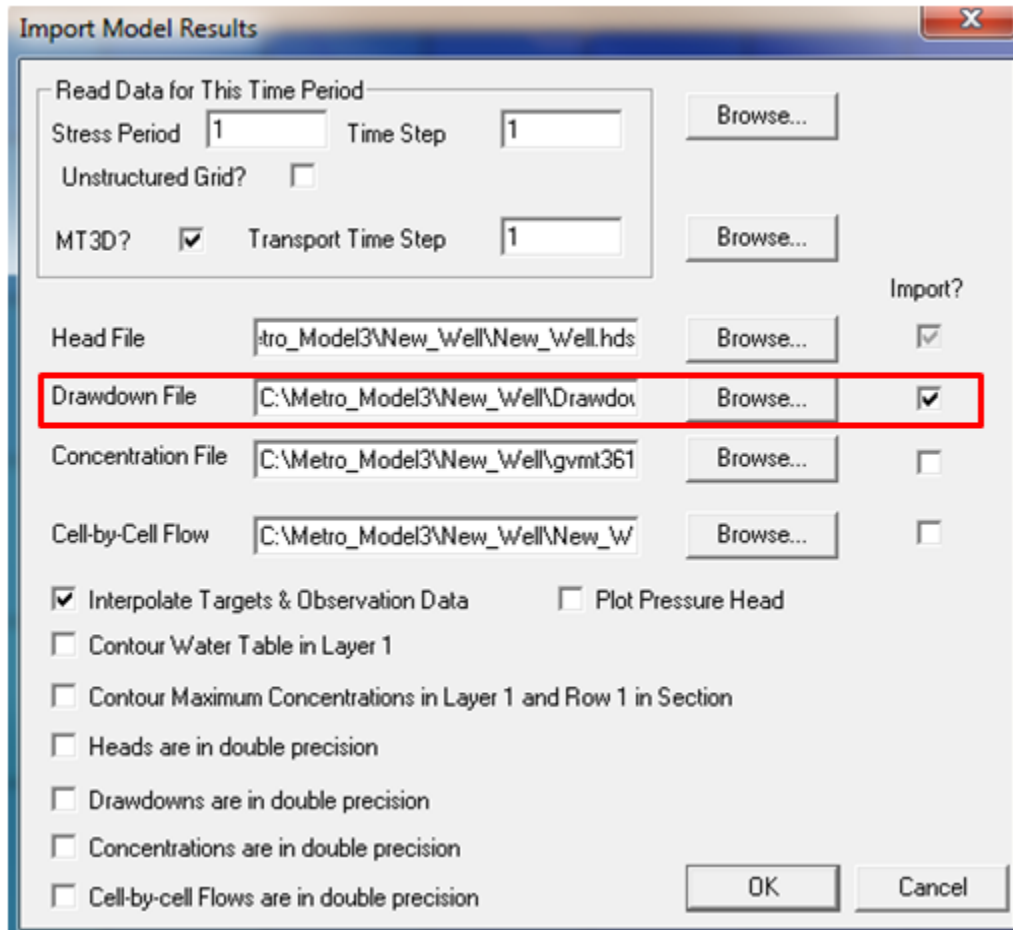


3. After the correct information is entered into the dialog window, select the **View** button to create the *Drawdown.ddn* file.

The *drawdown.ddn* file can now be read into Groundwater Vistas as a drawdown file for contouring and color-flooding.

4. Select **Plot | Import Results** and enter in or browse to the path of the new *drawdown.ddn* file in the Drawdown File dialog box. Select the **Import?** checkbox next to drawdown file and click **OK**.

Figure 2.7-2 Import Model Results dialog window

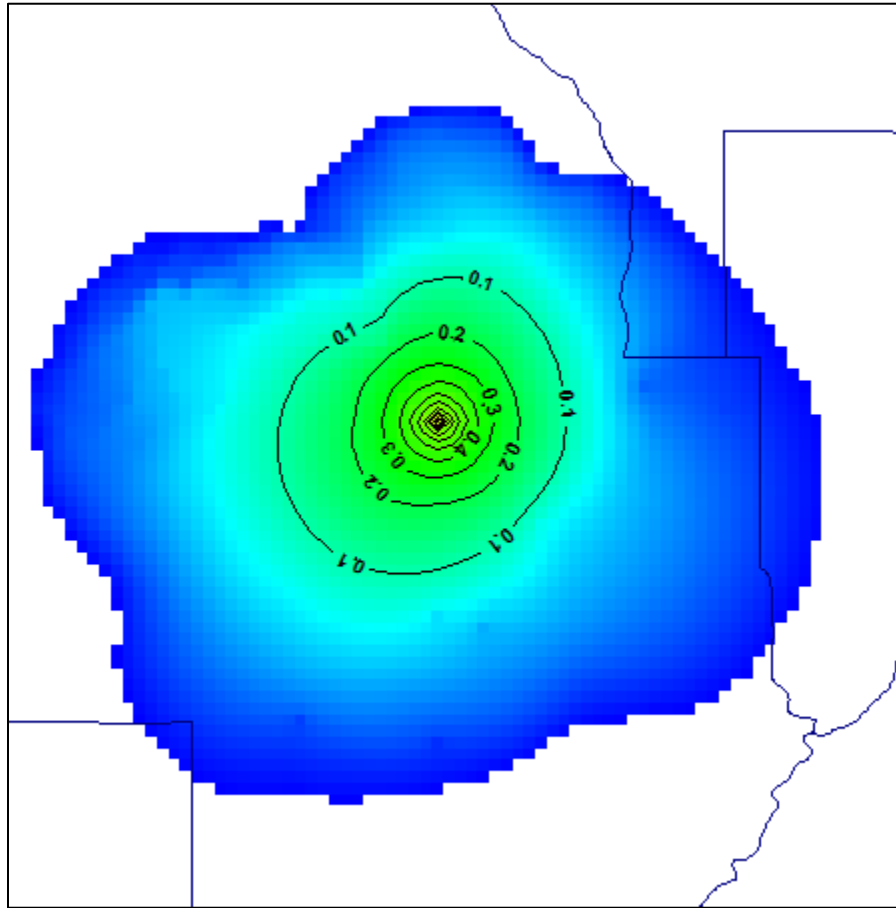


To visualize the amount of drawdown we need to adjust contouring and color flood setting in Groundwater Vistas:

5. Select **Plot | What to Display** and select Drawdown for both **Display Contours of** and **Display Color Flood of**. Make sure the check box is selected for both contours and color flood.
6. Select **Plot | Contours | Parameters (Plan)** and adjust the **Minimum Level** to -5 and the **Maximum Level** to 5. Adjust the **Interval** to 0.1. You may also want to adjust the contour color and labeling options.
7. Select **Plot | Color Flood | Color Flood Options** and set the **Minimum** to 0.01 and the **Maximum** to 10.0. Select the box for **Use Log Scale for Flooding** to make the colors easier to see. Unselect the checkbox for **Flood Areas Beyond Min and Max**.

Contours of drawdown at a 0.1m interval and color flooding of drawdown extending out to where drawdown is 0.01m will now be shown². Figure 2.7-3 shows the result for model layer 4

Figure 2.7-3 Drawdown in model layer 4 from hypothetical new well



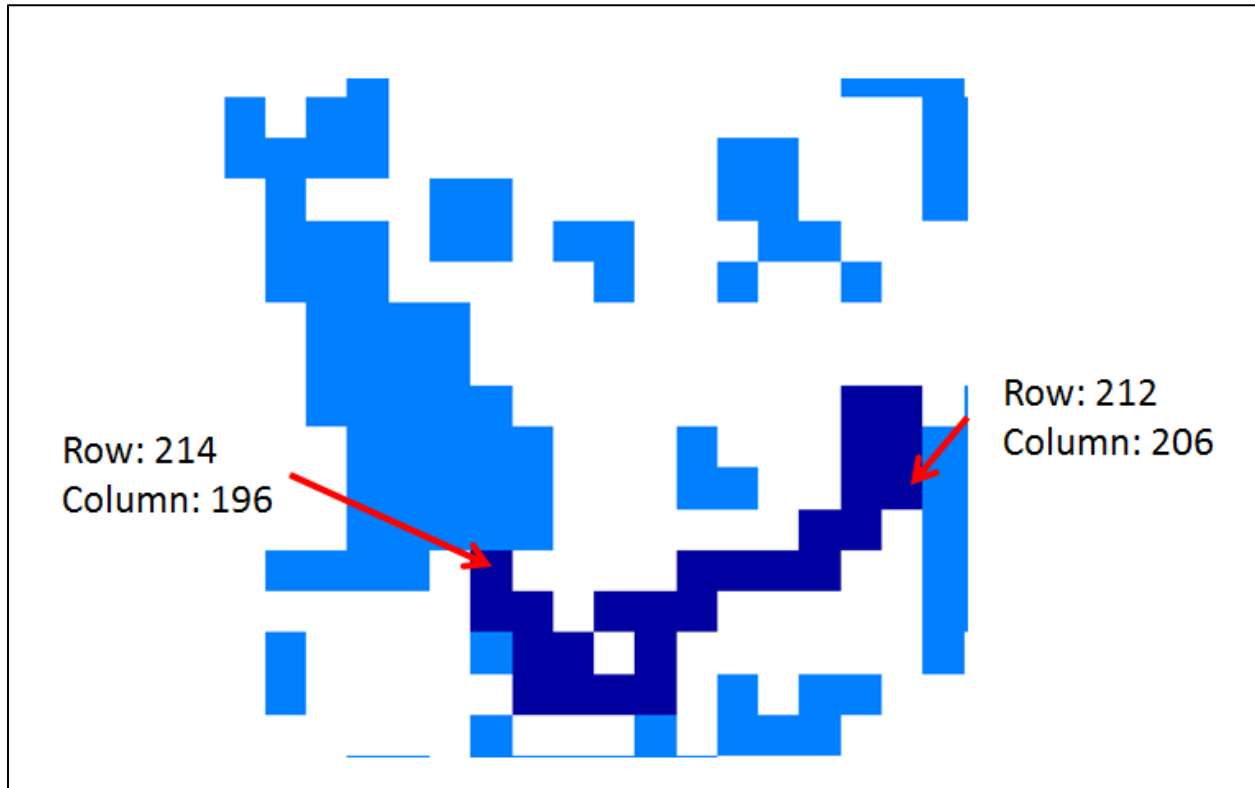
2.8 Evaluate Changes in Baseflow

To evaluate changes in baseflow, we need to define a reach we are interested in. We will assign reach numbers to river boundary cells representing Basset Creek between Medicine Lake and approximately six miles downstream of Medicine Lake.

1. Select **BCs | River** to set the river cell boundary active for modification.
2. **Double click** the river cell located at row 214, column 196. In the River Boundary Condition dialog box change the **Reach** number to 1234. Do this for all river cells shown in dark blue in Figure 2.8-1.

² In general, it is a good practice to not delineate a “zero” drawdown level because the numerical precision of the model will typically compute positive drawdown values far beyond where measurement methods could be used to detect them.

Figure 2.8-1 River cells to track changes in flux. Dark blue cells are river cells assigned to reach 1234 for this tutorial.



3. Select **Plot | Import results** and import results from the baseline simulation. Make sure the path to the cell-by-cell flow file is correct (e.g *baseline.cbb*) and the check box for **Import?** is selected.
4. Select **Plot | Mass Balance | Boundary Reach | Display One Reach** and enter reach number 1234. The simulated flux into reach 1234 will be displayed. The units for this flux are in m^3/day and the negative value indicates the water is leaving the model (a gaining reach). Note the flux value for comparison after completing Step 5
5. Repeat steps 3 and 4 from above for the new well simulation.
6. Comparing the simulated groundwater flux into reach 1234 for the two simulations indicates the amount of change in flux to the reach due to the addition of the new well.

Tutorial 3: Model Refinement and Capture Zone Analysis

3.1 Introduction




When assessing impacts associated with a well or determining capture zones using particle tracking, it is often necessary to refine the model grid around the well and/or area of interest. We will refine the model grid in an area around the new well added in Tutorial 2 to allow for more accurate particle tracking near the well. Grid refinement will also encompass Basset Creek and Medicine Lake and allow for more detail to be added for surface-water features in the area of interest. Additionally, we will manually adjust hydraulic conductivity values for an area around the new well.

3.2 Getting Started

This tutorial will build on Tutorial 2. Model files built during Tutorial 2 can be used as a starting point. The Groundwater Vistas file `.\MM3_User_Manual\Tutorials\Tutorial2\MM3_Steady_State_New_Well.gvw` can also be used as a starting point if Tutorial 2 was not completed. GIS files for mapping surface-water features to the refined model grid are provided in `.\MM3_User_Manual\Tutorials\Tutorial3\GIS_Files`. A Groundwater Vistas file derived from completing this tutorial is provided for reference at `.\MM3_User_Manual\Tutorials\Tutorial3\MM3_Steady_State_New_Well_Refined.gvw`

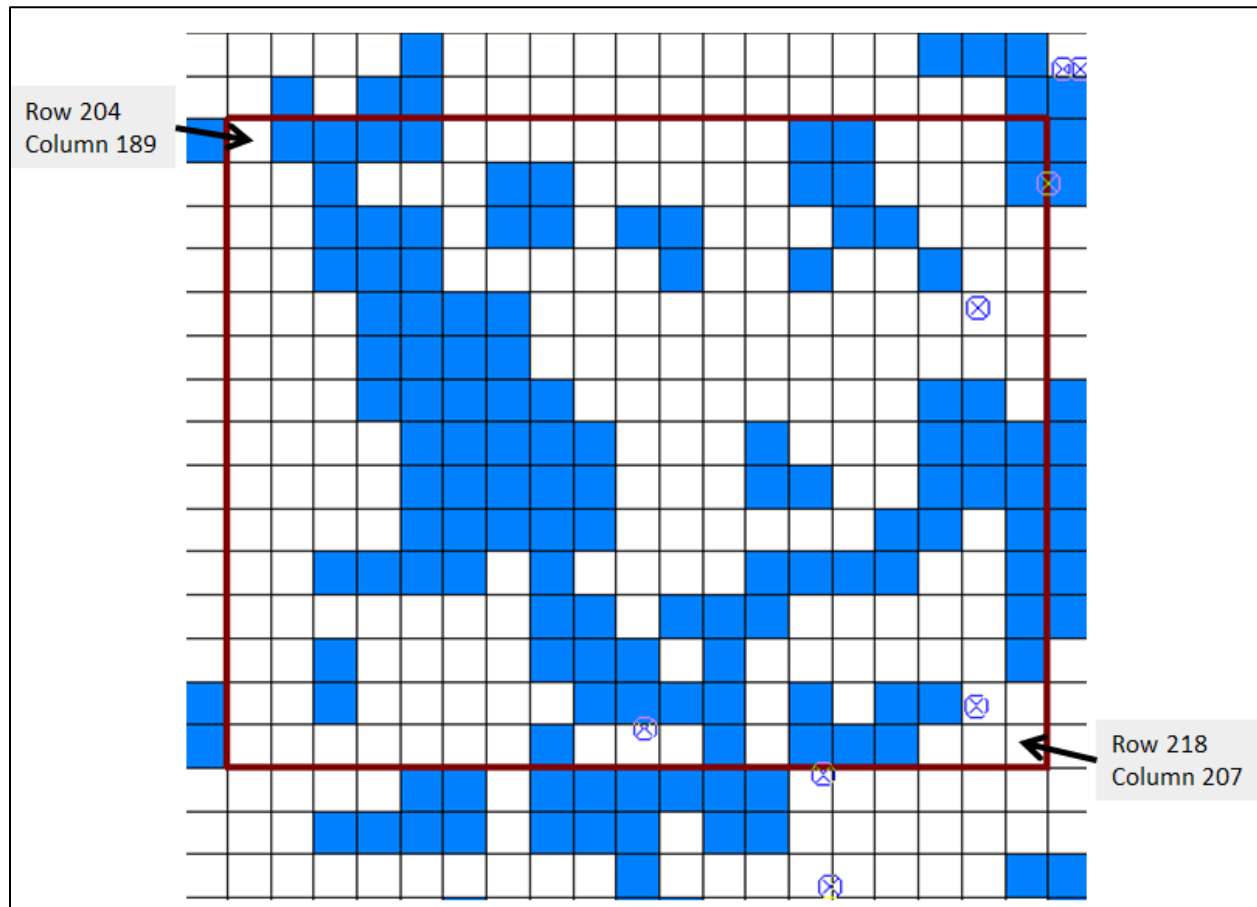
3.3 Refine Model Grid

There are several ways to refine the model grid within Groundwater Vistas. An automated option to refine the model grid is available; however, when used after boundary conditions have already been defined this method often causes those boundaries (e.g. river cells) to be displaced and conductance values will not remain consistent. We will manually add in rows and columns so that boundaries are not displaced and total conductance values remain consistent. Boundaries (river cells) within the area of grid refinement will be adjusted in Section 3.4.2.

1. Select the **edit grid** button 
2. Select **Grid | Options**. Under the heading *Option for Adding New Rows/Columns* select **Split Cell in half**.
3. Select the **insert column** button  then right click in a column that you want to split in half. Continue to do so until the model grid is refined down to at least 62.5m column spacing in the area of interest, as indicated by the red box in Figure 3.3-1. The original cell size is 500 m x 500 m. So, the first split results in 250 m column spacing, the second split in 125 m column spacing, and the third split in 62.5 m spacing³.
4. Select the **inset row** button  and split the rows until the row spacing is refined down to at least 62.5 m spacing in the area of interest.

³ In the future, use of the newly developed MODFLOW-USG (“USG” standing for “unstructured grid”) will likely result in changes to the way the grid in Metro Model 3 will be refined in particular areas of interest but MODFLOW-USG is in the early stages of use among practitioners and application to Metro Model 3 may be addressed at a later time.

Figure 3.3-1 Area of interest for grid refinement



It is a best practice to only change row or column spacing by less than 50 percent at a time along the edges of the refined area. See Figure 3.4-4.

3.4 Add Local Hydrogeologic Data

Now that the model grid is refined in the area of interest we will manually adjust aquifer parameters around the new well and import river boundary condition data to better represent Basset Creek, Medicine Lake, and other surface water features in the area of grid refinement.

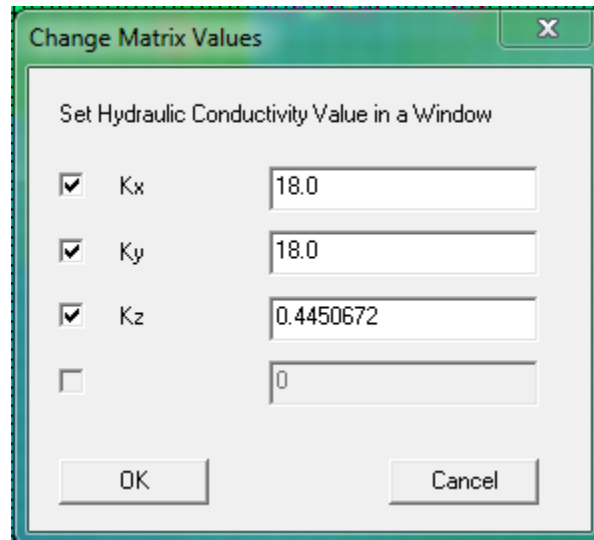
3.4.1 Manually Adjust Parameters

It is likely that local aquifer testing will provide hydraulic conductivity values that differ somewhat from those in the Metro Model 3. We will assume that results of a pumping test(s) at the site of the new well indicate that the horizontal hydraulic conductivity of the Prairie du Chien Group should be 18 m/day and the Jordan Sandstone should be 12 m/day. The calibration of Metro Model 3 involved a series of complicated processing steps to define hydraulic conductivity for the model. These included interpolating between pilot points and applying a depth dependent function. Use of these processing steps is beyond the scope of this tutorial. We will instead define an arbitrary area around the well to assign the desired hydraulic conductivity values.

1. Navigate to model layer 3
2. Select **Props | Hydraulic Conductivity** to activate the hydraulic conductivity property for editing.

3. Select **Props | Set Value or Zone | Window**. Draw a box around the new well.
4. The Change Matrix Values dialog window will pop up. Change Kx and Ky to 18.0 m/day.

Figure 3.4-1 Change Matrix Values dialog box



5. Navigate to model Layer 4
6. Select **Props | Set Value or Zone | Window**. Draw a box around the new well.
7. Change Kx and Ky to 12.0 m/day.

3.4.2 Import New/Updated Data from GIS

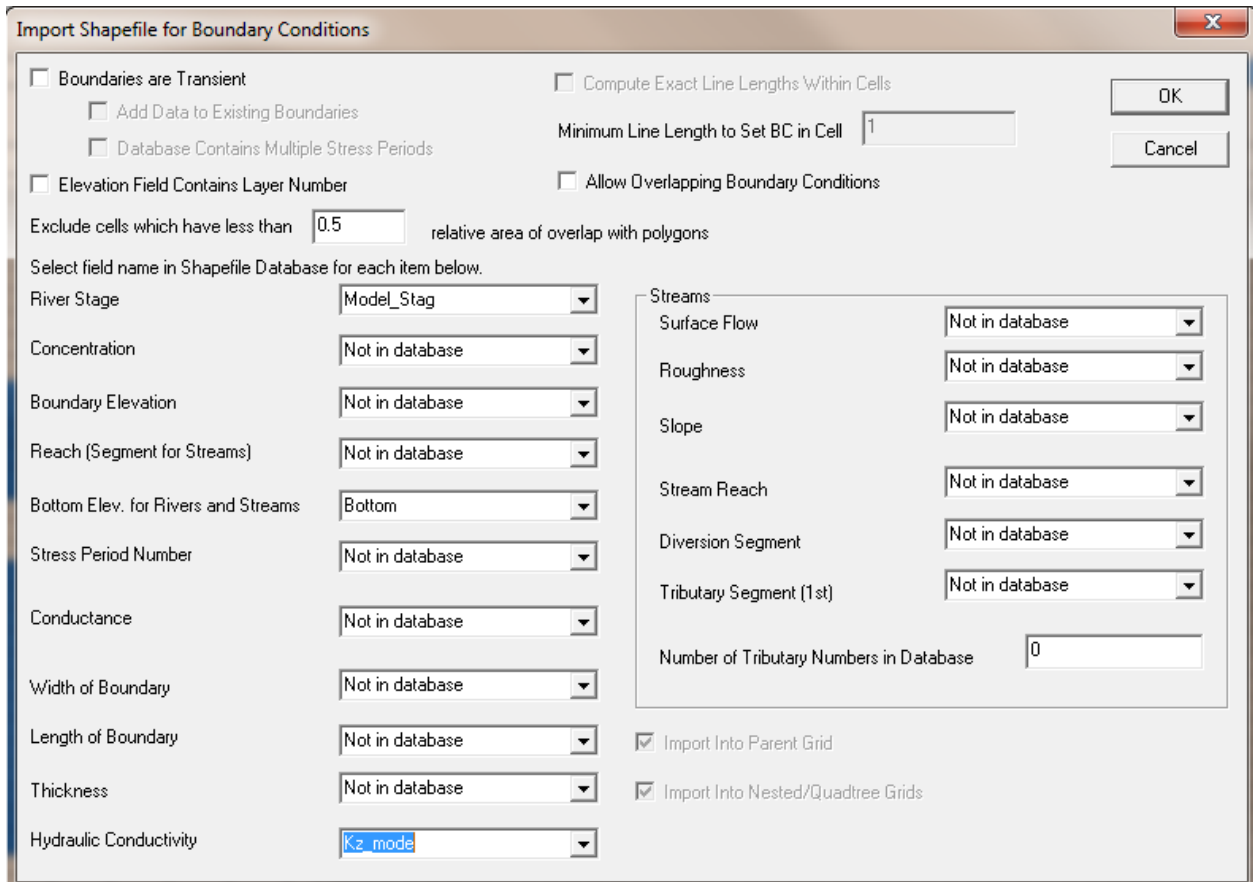
Now that the grid has been refined in the area of interest we will import GIS data to redefine river boundary conditions in order to better capture the geometry of surface-water features in the area of interest.

1. Navigate to model Layer 1
2. Select **BCs | River** to active the river boundary condition for editing
3. Select **BCs | Delete** and draw a window in the area of refinement to delete all River Boundary Condition cells within the window.

Shapefiles for lakes/ponds and Bassets Creek have been provided as part of the tutorial files. These files are similar to those provided for the entire model but have been clipped to the area of refinement to make adding in the data to the model easier.

4. Select **BCs | Import | Shapefile**
5. Navigate to the shapefile *Surface_Water_Features.shp* provided and select **Open**
6. In the Import Shapefile for Boundary Conditions dialog window, set a value to **0.5** for **Exclude cells which have less than 0.5 relative area of overlap with polygons**.
7. For **River Stage** select Model_Stag attribute field.
8. For **Bottom Elev. For Rivers and Streams** select Bottom attribute field.
9. For **Hydraulic Conductivity** select Kz_mode attribute field.
10. Select **OK** to import data.

Figure 3.4-2 Import Shapefile for Boundary Conditions dialog box



11. Select **BCs | Import | Shapefile**
12. Navigate to the shapefile *Basset_Creek.shp* and select **Open**
13. For the prompt 'Does Database Contain Data for Both Ends of Polyline' select **Yes**
14. In the Import Polyline Shapefile for Boundary Conditions dialog box. select options as shown in Figure 3.4-3. Groundwater Vistas will interpolate stage elevations to each model cell and will use the width, calculated line length within each cell, and hydraulic conductivity to determine a conductance for each river cell.
15. Select **OK** to import data for Basset Creek.

The updated river boundary cells should look similar to Figure 3.4-4

Figure 3.4-3 Import Polyline Shapefile for Boundary Conditions dialog box

Import Polyline Shapefile for Boundary Conditions

Boundaries are Transient

Add Data to Existing Boundaries

Database Contains Multiple Stress Periods

Elevation Field Contains Layer Number

Select field name in Shapefile Database for each item below.

	Beginning of Line	End of Line
River Stage	Stg_start	Stg_end
Concentration	Not in database	Not in database
Boundary Elevation	Not in database	Not in database
Reach (Segment for Streams)	Not in database	
Bottom for River Boundaries	Bot_start	Bot_end
Stress Period Number	Not in database	
Conductance	Not in database	Not in database
Width of Boundary	Width	Width
Length of Boundary	Not in database	Not in database
Thickness	Thick	Thick
Hydraulic Conductivity	Kz	Kz

Streams

Surface Flow: Not in database

Roughness: Not in database

Slope: Not in database

Diversion Segment: Not in database

Tributary Segment (1st): Not in database

Number of Tributary Numbers in Database: 0

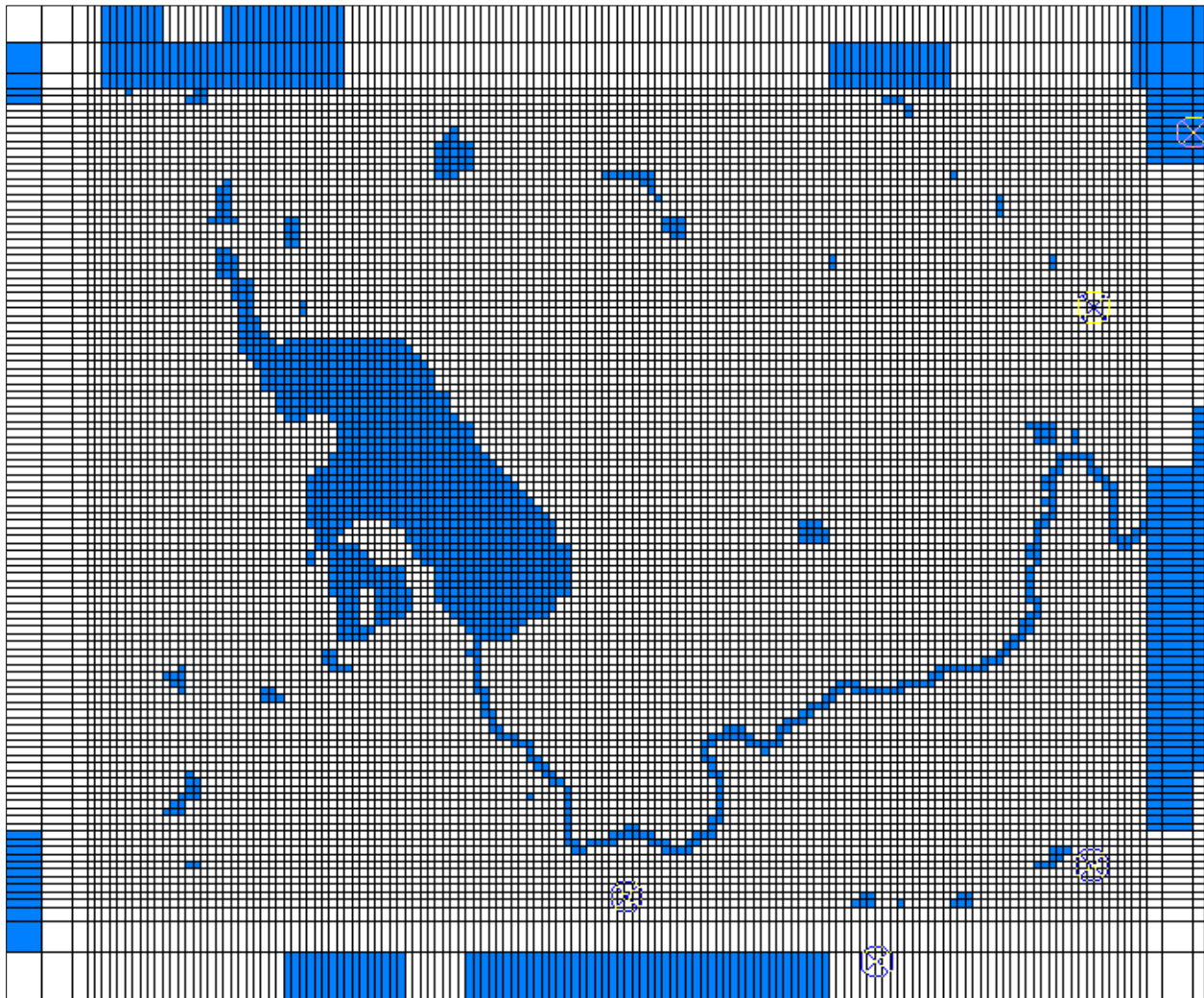
Import Into Parent Grid

Import Into Nested/Quadtree Grids

Compute Exact Line Lengths within Cells

Minimum Length to Set BC Cell: 1

Figure 3.4-4 Result of importing surface water features from GIS data for refined grid



3.5 Evaluate Drawdown and Well Interference

We can now run the model to assess impacts related to pumping. The process is nearly identical to the methods described in Tutorial 2. A brief summary of the process is described below.

1. Select the New Well (the analytic element) and set the pumping rate to 0 m³/day.
2. Select **Model** | **MODFLOW** | **Packages** and change the Root File Name to *Refined_Baseline* (or anything similar).
3. Generate the model files and run the model. The model will take a while to converge due to the increase in the total number of model cells.
4. Select the New Well and set the pumping rate to -2725 m³/day (500 gpm)
5. Select **Model** | **MODFLOW** | **Packages** and change the Root File Name to *New_Well_Refined* (or anything similar).
6. Generate the model files and run the model
7. Select **Plot** | **File Operations** | **Compare Two Files**.
8. Enter appropriate information into the Compare Files dialog window. We are subtracting the new well head file (*New_Well_Refined.hds*) from the baseline head file (*Refined_Baseline.hds*) to compute drawdown. Results can be saved into a new file called *Refined_Drawdown.ddn*.

9. After information is entered into the Compare Files dialog window select the **View** button to create the *Refined_Drawdown.ddn* file.

The *Refined_Drawdown.ddn* file can now be read into Groundwater Vistas as a drawdown file for contouring and color-flooding. See Tutorial 2 (Section 2.7) for more details.

10. Select **Plot | Import Results** and enter in or browse to the path of the new *Refined_Drawdown.ddn* file in the Drawdown File dialog box. Select the **Import** checkbox next to drawdown file
11. Select **Plot | What to Display** and select **Drawdown** for both **Display Contours of** and **Display Color Flood of**. Make sure the check box is selected for both contours and color flood.
12. Select **Plot | Contours | Parameters (Plan)** and adjust the **Minimum Level** to -5 and the **Maximum Level** to 5. Adjust the **Interval** to 0.1. You may also want to adjust the contour color and labeling options.
13. Select **Plot | Color Flood | Color Flood Options** and set the **Minimum** to 0.01 and the **Maximum** to 10.0. Select the box for **Use Log Scale for Flooding** to make the colors easier to see. Unselect the checkbox for **Flood Areas Beyond Min and Max**.

It should be noted that drawdown from simulations with the refined grid (as described in this tutorial) are not directly comparable to drawdown from Tutorial 2 because the hydraulic conductivity values were changed in Section 3.4.1.

There are many ways to assess well interference. Most commonly, drawdown contours associated with a new well are overlain on a map of existing wells. Only wells in the MN DNR SWUDS database are in Groundwater Vistas so we will export drawdown data as a GIS shapefile.

14. Navigate to layer 4
15. Select **File | Export**.
16. In the Export dialog window designate a **File name** of *Refined_Drawdown.shp* and **Save as type** as Shapefile then click **Save**.
17. In the Export Model Results to and ArcView Shapefile dialog window select Contours for **Result Type** and click **OK**. Note: drawdown contours will only be exported for the model layer currently being displayed. The coordinate system is not saved within the GIS file. However, it is conformable with UTM NAD 1983 Zone 15N.
18. The exported shapefile can now be read into GIS software and compared to the CWI well location shapefile (available from the Minnesota Geological Survey at <http://www.mnngs.umn.edu/cwi.html>).

Comparing wells in CWI to the 0.5 m drawdown contour for the Jordan aquifer reveals 1 well in the Jordan aquifer within the 0.5 m drawdown contour.

3.6 Assess Capture Zone of New Well using Particle Tracking

Particle tracking is commonly used to assess capture zones or groundwater-flow directions. Particle tracking considers only advective groundwater transport. Simulation of contaminant mass-transport that include dispersion, dilution, and chemical reactions is beyond the scope of this tutorial and Metro Model 3 will require substantial grid refinement for most contaminant-transport simulations. Particle tracking may be appropriate as a screening tool to determine the need for full contaminant-transport simulations for a particular site or problem.

MODPATH is the most commonly used software for particle tracking. We will use MODPATH to assess the 10-year capture zone for the New Well.

1. Navigate to model Layer 3 and zoom into the New Well.
2. Select **AE | Particle | Circle**
3. Hover the mouse pointer over the cell containing the New Well and left click. While holding the left mouse button down drag the pointer away from the well to create a circle.
4. In the Circle Trace Information dialog box, enter in data as shown in Figure 3.6-1.

Figure 3.6-1 Circle Trace information dialog box

The image shows a software dialog box titled "Circle Trace Information". It contains the following fields and options:

- Trace Information:** Radius (10), Number (10).
- Type:** Radio buttons for Streamline and Particle (selected).
- Spatial Parameters:** X (99725), Y (100225), Top Layer (3), Bottom Layer (4), Z Offset (1), Release Time (0).
- Multiple Particles:** Number of Vertical Release Points (10), Minimum Z Offset (0), Maximum Z Offset (1).
- Optimization of Managed Pumping:** Weight (1), Maximum Travel Time (10000000000).

Effective porosity values (referred to only as “porosity” in Groundwater Vistas) are required to calculate time-of-travel for particle tracking. Porosity is stored in the storage/porosity property in Groundwater Vistas. Porosity values were not part of the model calibration process but are included in Groundwater Vistas with assumed values for convenience. In general, porosity varies based on hydrostratigraphic units (accessible via the Hydrostratigraphy property) with the exception of model Layer 1. Values of porosity for each hydrostratigraphic unit are shown in Table 3.6-1. All porosity values in model Layer 1 are set at 0.25 and are used to define the saturated water content of the unsaturated zone for the UZF package. In general this is of little concern. However, parts of southern Washington County and northeast Dakota County have hydrostratigraphic units other than Quaternary sediments mapped to model Layer 1 (See Christianson and Wuolo, 2014). If conducting particle tracking in this part of the model the user should be careful in defining porosity values. In all cases, site-specific effective porosity values should be considered by the user.

Table 3.6-1 Porosity values

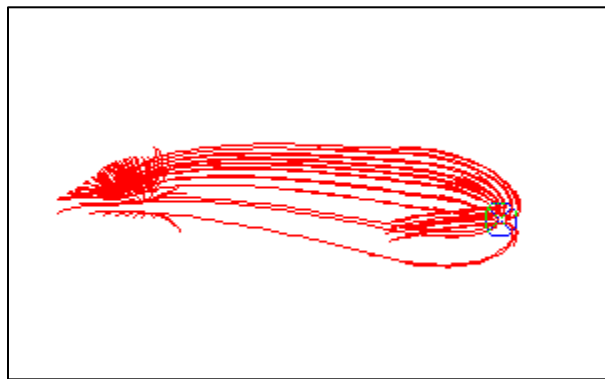
<i>Hydrostratigraphic Unit</i>	<i>Porosity</i>
Quaternary Sediments (undifferentiated)	0.25
St. Peter Sandstone	0.283
Prairie du Chien Group	0.056
Jordan Sandstone	0.318
St. Lawrence Formation	0.25
Tunnel City Group	0.25
Wonewoc Sandstone	0.253
Eau Claire Formation	0.25
Mt. Simon Sandstone	0.233
Cretaceous and unnamed Paleozoic	0.25

5. Select **Model | MODPATH | Packages** and change the Root File Name for MODPATH to *Refined_New_Well_MP*. This will help distinguish MODPATH and MODFLOW input and output files.
6. Select **Model | MODPATH | Particle Options**. In the MODPATH Particle Tracking Options dialog window, set **Direction of Tracking** to Reverse. Check the box for **Limit Particle Travel Time** and set the **Max. Travel Time** to 3650 days (10 years).

MODPATH needs to be informed about Quasi-3D layers used in the model. For additional details regarding Quasi-3D layers used in Metro Model 3 see Christianson and Wuolo, 2014.

7. Select **Model | MODPATH | Layer Geometry**. Set the **number of Quasi 3D layers** to 8 (Layer 9 does not have a Quasi-3D layer because it is the lower-most layer in the model)
8. Select **Model | MODPATH | Confining Beds**. Make sure “Yes” is selected for Layers 1-8, the thickness for each is set to 0.1, and the porosity is set to 0.05 (assumed value)
9. Select **Model | MODPATH | Create Datasets**
10. Select **Model | MODPATH | Run MODPATH**
11. After MODPATH has finished running, open the newly created pathline file (*.ptl) if prompted. If not prompted automatically, select **Plot | Import MODPATH** and select the pathline file for import.
12. If particles are now shown after importing the pathline file, select **Plot | What to Display** and check the box for **Display Particle Traces**

Figure 3.6-2 Ten-year reverse particle tracking from the New Well



Tutorial 4: Implementing Lake Package

4.1 Introduction

Tutorial 4 builds on Tutorial 3 by using the Lake Package to simulate the effects of pumping on water levels in a lake. All rivers and lakes in Metro Model 3 are simulated with the River Package. The River Package simulates water flow into or out-of the aquifer system using a defined stage for the river/lake and a conductance value representing river/lakebed sediments. The River Package implicitly assumes that river/lake stage does not vary with leakage into or out-of a river/lake in response to stresses (pumping, climate, etc.). The Lake Package calculates a lake stage based on a lake water budget and geometry of the lake. Water exchanged between the lake and aquifer, along with direct precipitation, evaporation, and stream inflow/outflow can be tracked to establish a lake water budget. Changes in seepage from a lake associated with increased pumping can be related to a change in lake stage.

4.2 Getting Started

This tutorial builds on Tutorial 3. Model files built during Tutorial 3 can be used as a starting point. Alternatively, a Groundwater Vistas file is provided to use as a starting point if Tutorial 3 has not been completed at `.\MM3_User_Manual\Tutorials\Tutorial3\MM3_Steady_State_New_Well_Refined.gvw`. GIS files used in this tutorial are provided at `.\Tutorial4\GIS_Files`.

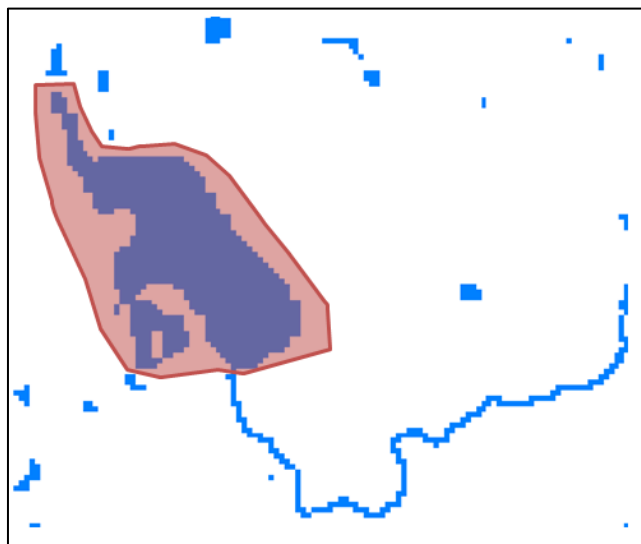
The file `.\Tutorial4\MM3_Steady_State_New_Well_Lake_Package.gvw` is a model derived from completion of this tutorial and is provided for reference.

4.3 Implementing the Lake Package

For illustrative purposes only, we will simulate Medicine Lake with the lake package and run the model with and without pumping from a hypothetical new well to estimate changes in lake stage.

1. Select **BCs | River** to activate the river boundary for editing.
2. Select **BCs | Delete | Polygon**. Draw a polygon to delete all river boundary cells that represent Medicine Lake shown in Figure 4.3-1.

Figure 4.3-1 Medicine Lake cells



3. Select **BCs | Lake** to active the lake boundary for editing.
4. Select **BCs | Import | Shapefile**. Browse to the shapefile of Medicine Lake provided and select **Open**.
5. In the *Import Shapefile for Boundary Conditions* dialog window select fields, as shown in Figure 4.3-2.

Figure 4.3-2 Import Shapefile for Boundary Conditions dialog box – Lake Boundary

Note that all lake cells will be in model Layer 1. The maximum depth of Medicine Lake is between 40 and 50 feet. The thickness of model Layer 1 is approximately 180 feet (55 m) near Medicine Lake. Lake geometry is generally represented by the lake-boundary cell-dimensions. It may be appropriate to add additional model layers to reduce the thickness of Layer 1 and better capture the geometry of the lake. However, introducing additional model layers is well beyond the scope of this tutorial. Alternatively, a new option in the lake package allows for the lake geometry to be defined in a separate file. However, this is a relatively new option and not available in Groundwater Vistas at the time of this writing.

6. Select **Model | MODFLOW2000 | Packages**. Check the box for **create Lake3 (LAK) Package** and assign a unit number of 21 and a Cell-by-Cell Flow Unit Number of 50. Any value between 2 and 900 can be assigned for the unit number of the LAK package. However, the same unit number cannot be assigned to multiple packages.

Direct precipitation onto the lake is implemented with data stored in the recharge property in Groundwater Vistas.

7. Select **Props | Recharge** to activate the recharge property for editing.
8. Select **Props | Set Value or Zone | Digitize Polygon**. Draw a polygon encompassing all lake-boundary cells. Enter a recharge (precipitation) value of 0.00202 m/day (29 in/yr).

Evaporation from the lake is implemented with data stored in the evapotranspiration package in Groundwater Vistas.


9. Select **Props | Evapotranspiration** to activate the evapotranspiration property for editing.
10. Select **Props | Set Value or Zone | Digitize Polygon**. Draw a polygon encompassing all lake-boundary cells. Enter an ET value of 0.00188 m/day (27 in/yr). If the user has set up ET to use zones, this will work as long as the zone covering Medicine Lake is set at 0.00188 m/day and the other zone(s) are 0.0 m/day

For the purpose of this tutorial, we will not simulate surface runoff into or out-of the lake. Implementation of these options often requires much more detailed analysis and may often require a surface-water routing model.

4.4 Evaluating Lake Levels

We will now assess the potential change in lake level associated with pumping a new hypothetical well. The new well is the same as was added to the model in Tutorials 2 and 3, located east of Medicine Lake in model Layers 3 and 4.

We will first run the model with the new well turned off (pumping rate of 0) to establish a simulated baseline lake level. We will then increase the pumping rate to 2,725 m³/day (500 gpm) and then 5,451 m³/day (1000 gpm) to assess potential changes in lake level associated with pumping the new well.

1. Navigate to model Layer 3 and select the new well. In the Well Information dialog window, enter a pumping rate of 0.
2. Create model datasets and run the model by clicking the **Recalculate** or **Run Model** button 
3. After the model calculations are complete, import the results (*.hds file) and hover the mouse over Medicine Lake (you may need to navigate back to model Layer 1). The simulated lake stage will be shown on the right side of the status bar (located at the bottom of the screen) as the cursor is moved. Note this simulated stage for future reference. The simulated lake stage may not equal the stage defined for the lake when importing data. This is because the stage is calculated based on the water budget of the lake. To better match actual lake stage the model would need to be calibrated by adjusting lake-bed conductance parameters and aquifer parameters.
4. Repeat steps 1-3 above, adjusting the pumping rate to -2,725 m³/day and -5,451 m³/day. Compare the calculated lake stage for both pumping conditions to the baseline (0 pumping at the New Well) lake stage.

Simulations run in development of this tutorial indicate a reduction in lake stage of 0.09 m (0.30 ft) when pumping at 500 gpm and a reduction in lake stage of 0.17 m (0.56 ft) when pumping at 1000 gpm. It should be noted that the model was set up with no surface runoff or outflow from the lake. Also, additional model layers may be warranted to better capture lake geometry. Including these would most certainly lead to a different result.

Tutorial 5: Modifying SWB Model

5.1 Introduction

Two of the most common questions regarding regional infiltration (groundwater recharge) are: (1) What effect does a change in land use have on infiltration? and (2) How does a change in climate (long-term climate change, or drought) affect infiltration? This tutorial is designed to guide the user on how to use SWB to assess these types of questions.

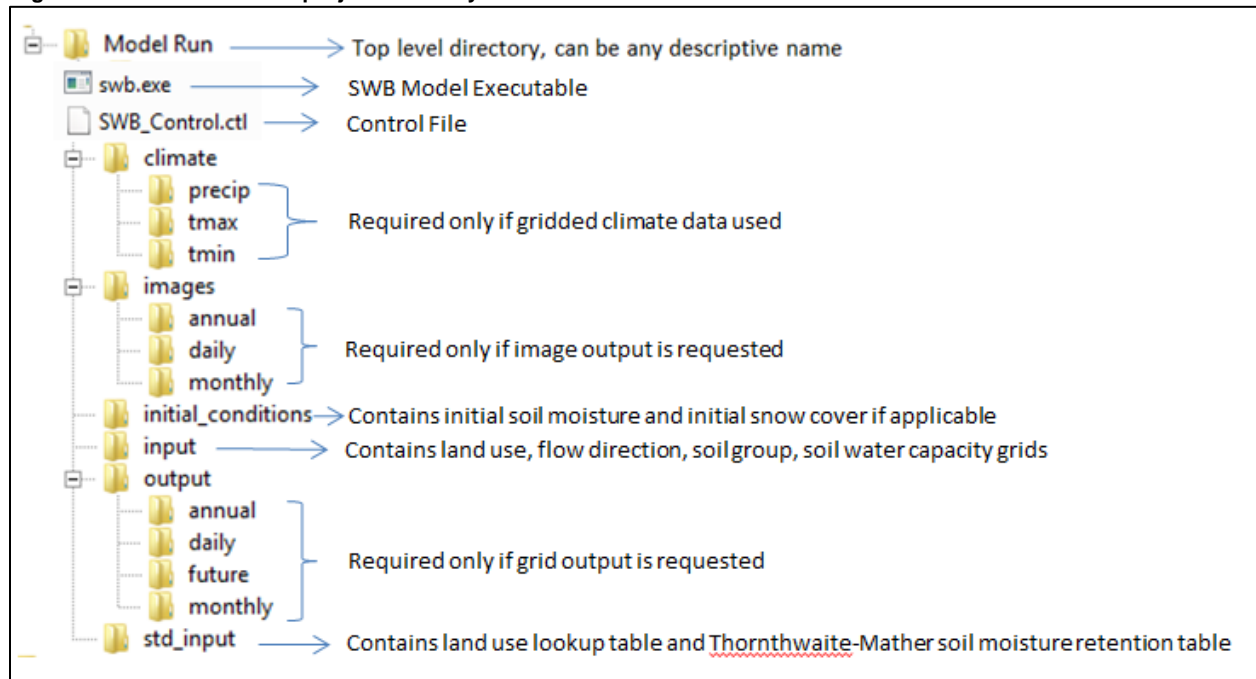
Three SWB simulations will be developed, run, and compared: (1) a baseline SWB run to use for comparison; (2) a simulation of climate change using aggregate results from a general circulation model; and (3) a simulation of a land-use change from rural row crop and forest to urban development.

5.2 Getting Started

Files associated with this tutorial are provided at `.\MM3_User_Manual\Tutorials\Tutorial5`. An example project directory structure is provided at `.\Tutorial5\SWB_Project_Directory`. Completed simulations as described are provided at `.\Tutorial5\SWB_Tutorial`. Additional reference files and utility scripts are provide at `.\Tutorial5\Misc`

SWB requires a standard project directory structure for input files and storing output. The general required directory structure is shown in Figure 5.2-1.

Figure 5.2-1 SWB standard project directory structure



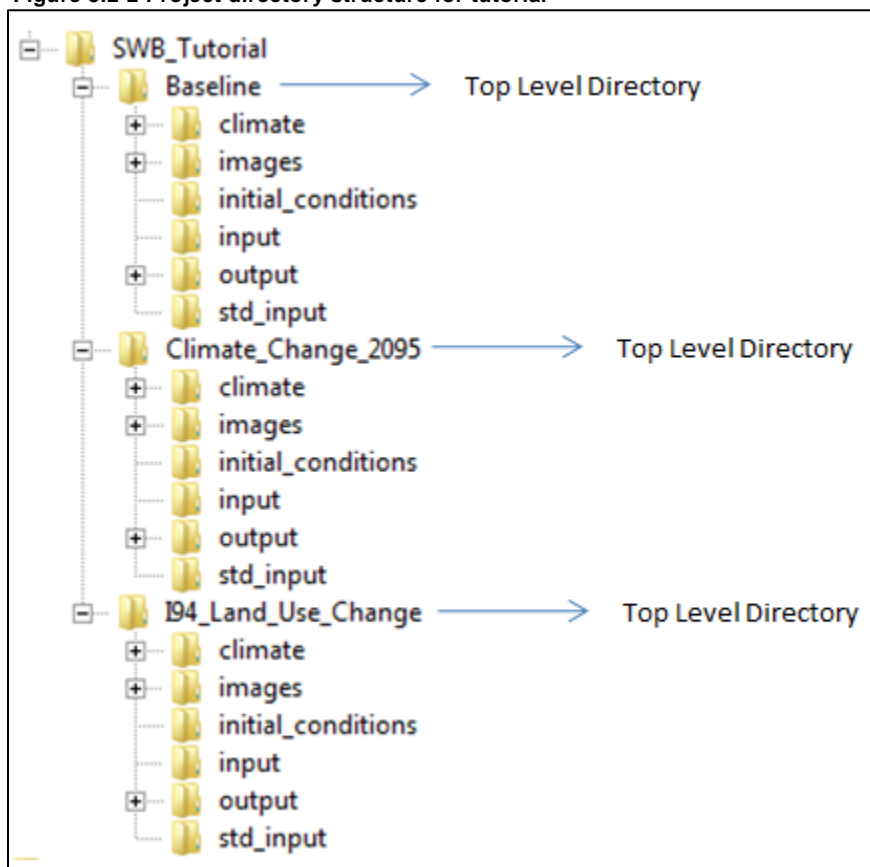
Input files and simulation options for SWB are designated using a control file. Each required input and option has a control-file statement followed by input values or relative file path. An example control file is provided in *SWB_Control.ctl*. Specific modifications to the control file pertinent to this tutorial are described. The main SWB manual (Westenbroek et al., 2010) should be referenced for all other options and input statements.

1. Copy the file directory *SWB_Project_Directory* to any convenient location; for best performance, the project directory should be copied to a local drive.

The project directory contains all the basic data and setup to run SWB. For this tutorial three separate SWB model runs will be completed and compared; (1) baseline, (2) climate change 2095, and (3) I94 land use change. All three, while based on realistic conditions, should be considered hypothetical examples for illustrative purposes only.

2. Copy the project directory two times and rename the top level directory as follows.

Figure 5.2-2 Project directory structure for tutorial



5.3 Modify Climate Data

For this tutorial two different climate datasets will be used: (1) an initial baseline dataset representative of current conditions and (2) simulation of climate change based on a composite of six general circulation models (GCMs).

The SWB model used for development of Metro Model 3 employs daily grid files for precipitation, maximum temperature, and minimum temperature for the period 1988-2011. These rasters were generated by interpolating data recorded at 191 weather stations. Using gridded climate input allows for spatial variability (particularly in regard to precipitation) to be accounted for in the calculation of infiltration. However, this approach also requires extremely large and difficult to manage input files (>1 TB). For the purpose of this tutorial, a single climatic record will be applied to all portions of the model domain. This helps alleviate file management issues for users of this tutorial and is more appropriate for simulations when comparing hypothetical changes in climate or land use.

5.3.1 Generate Baseline Synthetic Climate Data Set

The MarkSim DSSAT weather generator (Jones et al., 2011) will be used to create a baseline synthetic climate dataset. MarkSim DSSAT is employed here because of its ease-of-use but it is not endorsed for scientific study without further review. The general term for using large-scale climate data in local or regional models is “downscaling”.

1. Navigate to the MarkSim web interface at <http://gismap.ciat.cgiar.org/MarkSimGCM/>.

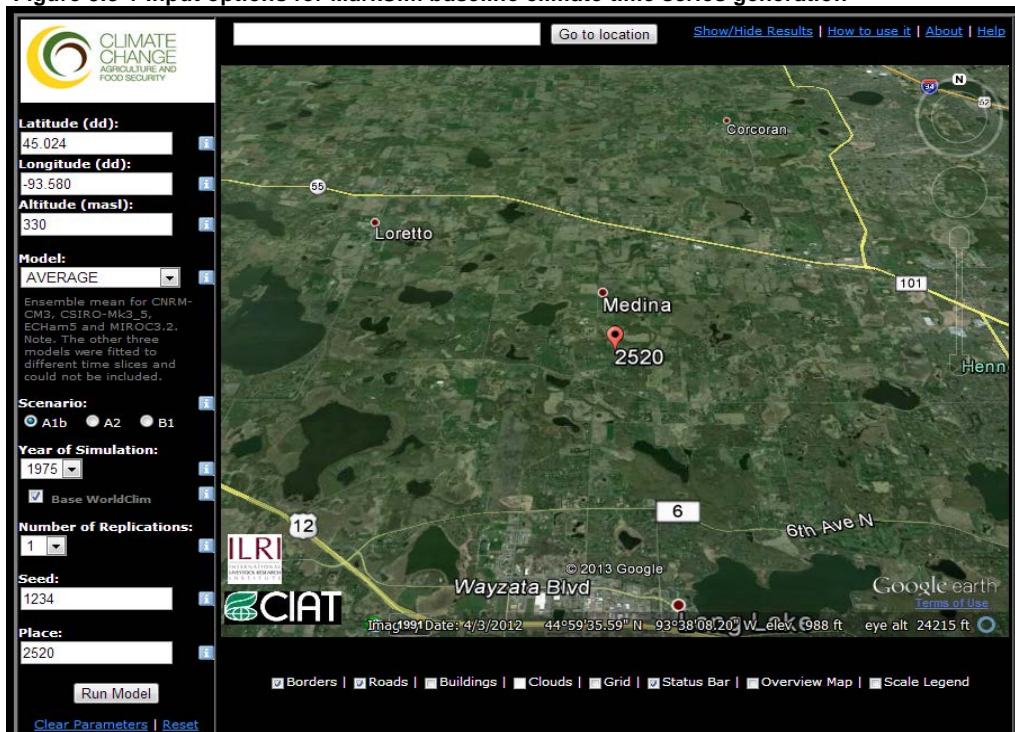
The MarkSim web interface offers a familiar Google Earth interactive environment for choosing a location for generated weather data.

2. Zoom into to the Twin Cities metro area and click a point on the map to select a location.

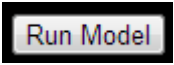
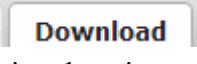
MarkSim will use the coordinates of the selected point to generate a weather time series. Alternatively, you can enter latitude and longitude data in the appropriate boxes. For reference, the center of the Twin Cities metro SWB model domain is located at latitude 45.024 and longitude -93.580.

3. Select options as shown in Figure 5.3-1.

Figure 5.3-1 Input options for MarkSim baseline climate time series generation



With options selected as shown above, a weather time series will be stochastically generated that is representative of baseline conditions (average of 1961-1990). A seed number of 1234 is entered to allow for comparable sets to be generated for different climate models or locations, if desired. The number of replications is set to 1. This means a single set (1 year) of climate data will be generated. If more rigorous statistical analysis is desired, many stochastic replications should be generated and SWB run and analyzed with each set.

4. Click on the  button.
5. Once the model has completed, click the  tab at the bottom of the screen, download the .zip file, and unzip the file to a convenient location.

Note: during development of this tutorial it was noted that at times downloading the data can take a long time (more than a few minutes). This appears to be sporadic and if it occurs simply try again at a later time. Data are also provided with this tutorial at:

`.\MM3_User_Manual\Tutorials\Tutorial\Misc\MarkSim_output`

5.3.2 Convert Baseline Synthetic Data Set to SWB Climate Time Series Format

The downloaded file will be named *CLIM0101.WTG*, etc. or similar. This is a simple ASCII text file and can be opened in any program that can read such files (i.e. notepad on most Windows machines).

Figure 5.3-2 Example of MarkSim output

*WEATHER DATA :						
@ INSI	LAT	LONG	ELEV	TAV	AMP	REFHT WNDHT
CLIM	45.024	-93.580	309			
@DATE	SRAD	TMAX	TMIN	RAIN		
01001	4.1	3.1	-11.1	0.0		
01002	2.9	6.3	-3.9	0.0		
01003	0.4	-6.1	-8.7	0.5		
01004	6.3	-12.1	-15.0	0.0		
01005	6.2	-12.8	-13.6	0.0		

The time-series data in the MarkSim output file needs to be converted to a time-series climate data file suitable for input to the SWB model. The SWB climate time-series file should be tab-delimited with all fields as shown in Figure 5.3-3. The MarkSim output assigns arbitrary dates for each day; SWB requires actual dates. For the baseline climate data a year of 1975 can be assigned. Note that all fields are not required unless alternative evapotranspiration methods are used. If a field is not used, it is convention to fill those values with -99999; blank values are not acceptable. Also, the MarkSim output is in units of millimeters for precipitation and degrees Celsius for temperature. These need to be converted to inches and degrees Fahrenheit for SWB input. The average temperature is assumed to be the mean of the minimum and maximum temperatures.

1. Convert the MarkSim output to the **tab delimited** format as shown in Figure 5.3-3. Standard spreadsheet programs such as Microsoft Excel can be used to correctly format this file. Alternatively, a python script is also provided that will automatically convert the MarkSim output to SWB format (Python 2.x, not compatible with Python 3.x). This script is located at `|SWB_Tutorial\Misc\MarkSim_output\MarkSim2SWB.py`.

Figure 5.3-3 Example of SWB climate time series input file for baseline scenario

Month	Day	Year	AvgTemp (F)	Precip (in)	AvgRelHum (%)	MaxTemp (F)	MinTemp (F)	Wind (m/sec)	MinRelHum (%)	Sunshine (%)
1	1	1975	24.80	0.00	-99999	37.58	12.02	-99999	-99999	-99999
1	2	1975	34.16	0.00	-99999	43.34	24.98	-99999	-99999	-99999
1	3	1975	18.68	0.02	-99999	21.02	16.34	-99999	-99999	-99999
1	4	1975	7.61	0.00	-99999	10.22	5.00	-99999	-99999	-99999
1	5	1975	8.24	0.00	-99999	8.96	7.52	-99999	-99999	-99999
1	6	1975	-3.37	0.00	-99999	8.06	-14.80	-99999	-99999	-99999

This SWB climate time series will be used for the baseline simulation and the I94 land use change simulation. It should be saved as an ASCII text file (tab delimited).

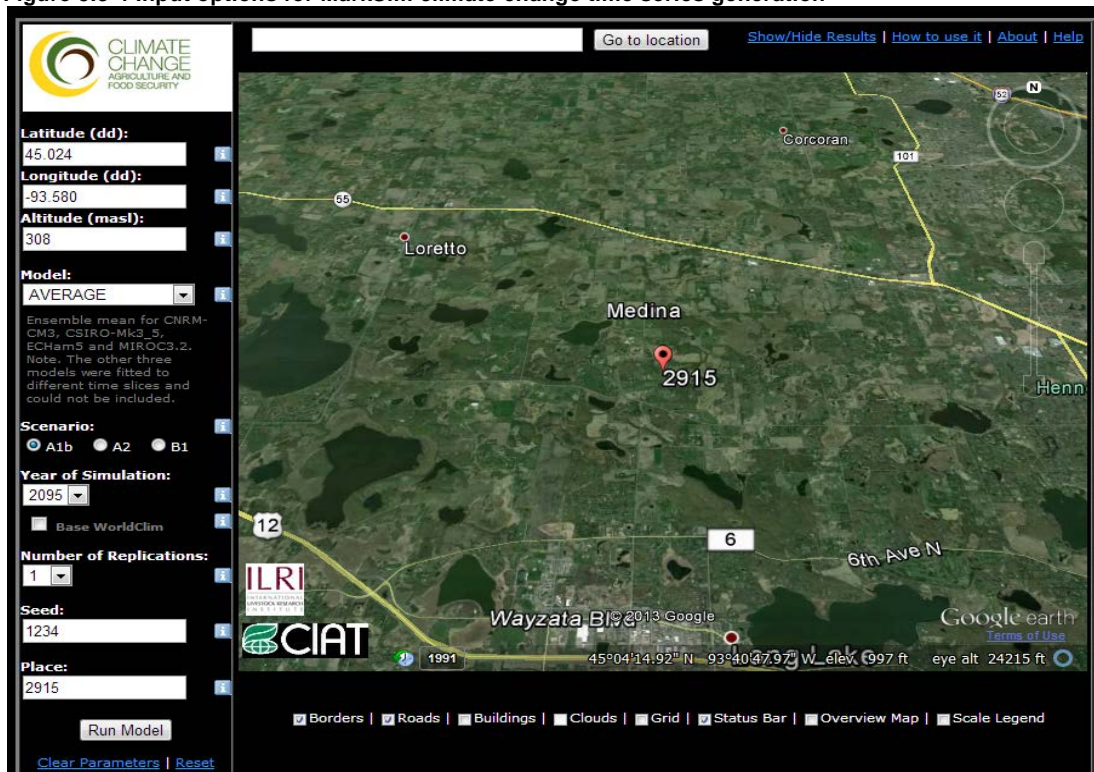
2. Save the SWB climate time-series file to the baseline project directory at **\Baseline\climate\1975.txt** and to the I94 land use change project directory at **\I94_Land_Use_Change\climate\1975.txt**

5.3.3 Generate Climate Change Synthetic Data Set

Similar to the baseline conditions, the MarkSim DSSAT weather generator can be used to generate a synthetic climate dataset using several different general circulation models representative of future climate conditions. Downloading the climate-change dataset is nearly identical to the steps followed in Section 5.3.1. The only differences are which model to use and representative year to simulate. For detailed instructions refer to Section 5.3.1.

1. Download a MarkSim climate time series using options as shown in Figure 5.3-4.

Figure 5.3-4 Input options for MarkSim climate change time series generation



With options selected as shown above a weather time series will be stochastically generated that is representative of the average of six different GCM models. A seed number of 1234 is used again to allow for comparable climate series. However, if more rigorous statistical analysis is desired many stochastic replications should be generated and SWB run and analyzed with each set.

5.3.4 Convert Climate Change Synthetic Data Set to SWB Climate Time Series Format

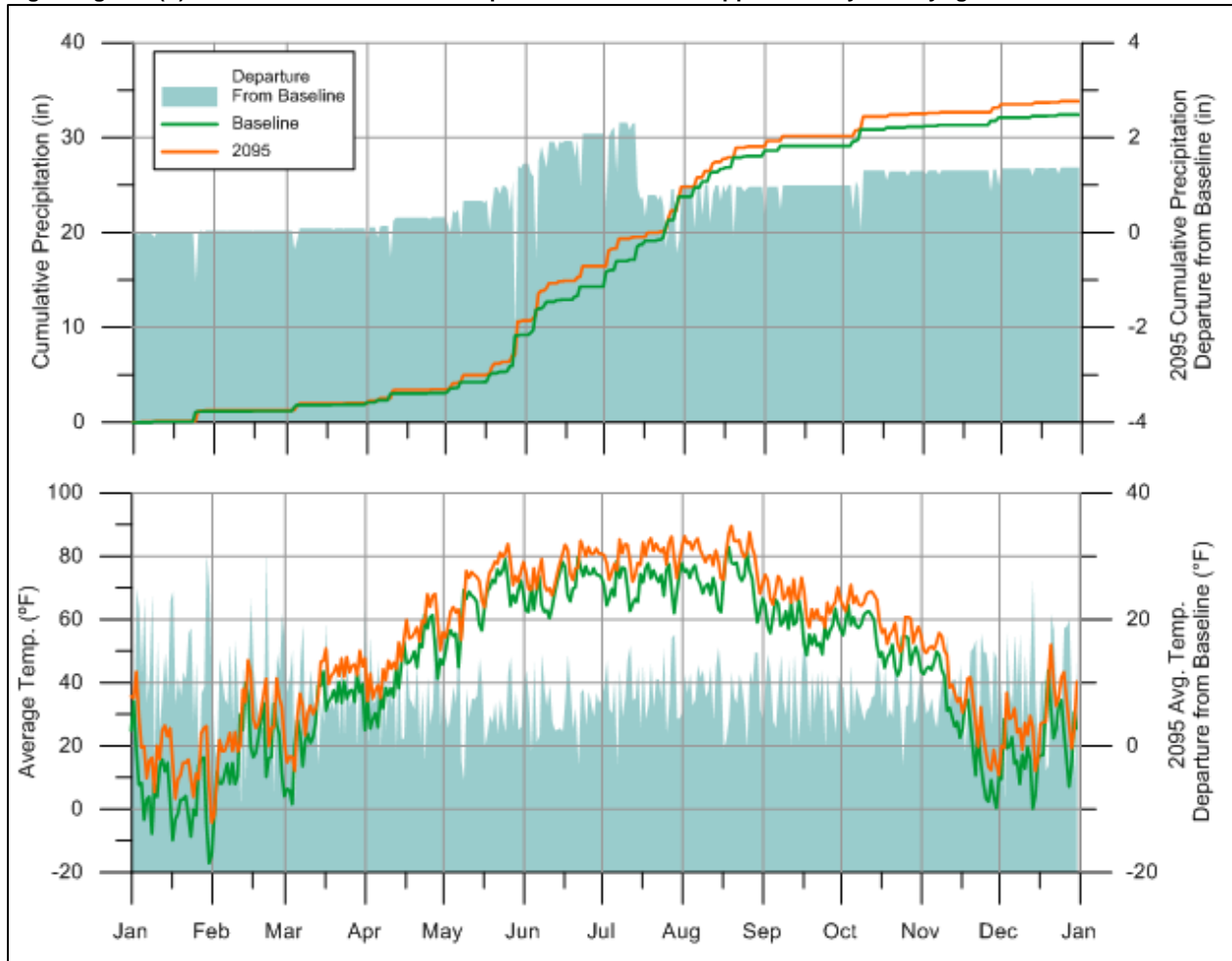
1. Convert the MarkSim output to the **tab delimited** format, as shown in Figure 5.3-5. For additional instruction, refer to Section 5.4.2.

Figure 5.3-5 Example of SWB climate time series input file for climate change scenario

Month	Day	Year	AvgTemp (F)	Precip (in)	AvgRelHum (%)	MaxTemp (F)	MinTemp (F)	Wind (m/sec)	MinRelHum (%)	Sunshine (%)
1	1	2095	35.69	0.00	-99999	49.64	21.74	-99999	-99999	-99999
1	2	2095	35.06	0.00	-99999	46.40	23.72	-99999	-99999	-99999
1	3	2095	43.43	0.00	-99999	51.80	35.06	-99999	-99999	-99999
1	4	2095	29.30	0.02	-99999	31.10	27.50	-99999	-99999	-99999
1	5	2095	19.49	0.00	-99999	21.38	17.60	-99999	-99999	-99999
1	6	2095	19.85	0.00	-99999	20.12	19.58	-99999	-99999	-99999

2. Save the SWB climate time-series file as an ASCII text file (tab delimited) to the climate change 2095 project directory at `\Climate_Change_2095\climate\2095.txt`.

Figure 5.3-6 Comparison of baseline and 2095 climate time series data. Temperatures in 2095 simulated approximately eight degrees (F) warmer than baseline. Precipitation simulated at approximately 1.5 in/yr greater



We now have data ready to simulate the baseline and 2095 simulations. However, we will first set up data for the land-use/land-cover change simulation. Skip to Section 5.5 if you don't plan on setting up the land-use/ land-cover change simulation.

5.4 Modify Land Use/Land Cover

To change the geographic distribution of land use/land cover used by SWB the integer raster grid of land use must be modified. Integer values in the land-use grid are used to lookup curve-numbers, interception, maximum-recharge, and rooting-depth in the land-use lookup-table located in the `\std_input` folder of the project directory. Current land-use classification and their corresponding integer lookup-values are as follows:

Table 5.4-1 SWB land use categories and associated integer ID values

<i>ID</i>	<i>Description</i>	<i>ID</i>	<i>Description</i>
11	Open Water	43	Mixed Forest
21	Low Density Residential	51	Shrubland
22	High Density Residential	71	Grass/Herbs
23	Commercial Industrial Transport	81	Pastures
31	Bare Rock	82	Row Crops
32	Quarries and Pits	85	Urban/Recreational Grasses
41	Deciduous Forest	92	Wetland
42	Evergreen Forest		

For this tutorial we will modify the existing land-use grid using an overly simplified and generic land-use change of urban growth along the Interstate-94 corridor. Existing land-use of mostly row crops and forest will be replaced with commercial/industrial/transportation (lookup value 23) near the interstate and low density residential further from the Interstate (lookup value 21).

If a new land-use category is desired (not listed above), the land-use grid should be modified as described below; but a new, unique, integer representing the land use should be added to the land-use lookup-table in the `\std_input` folder. For additional details, refer to the SWB user manual.

To modify the land-use grid for this tutorial, ESRI ArcMap will be used. There are likely several methods to reach the same end-product using ArcMap. This tutorial goes through what is believed to be the most common workflow. It is assumed that the user is familiar with basic tasks such as modifying/creating a shapefile and running standard tools from the ArcToolbox. There are numerous online tutorials and instructions, including the ArcMap help for learning how to do so. To complete this tutorial a license for spatial analyst is necessary. Alternative software such as Surfer, Numerical Python (Numpy), or manual modification of the raw ASCII grid could also be used. The best method to use will be dependent on available data and user's familiarity with the software and data structure.

1. Open ArcMap and create a polygon shapefile of the extent of the land-use change using the standard editing tools. The attribute table of this shapefile should contain an integer field to define the land use.

Figure 5.4-1 Land use; (A) existing land use and (B) hypothetical land use change along I94 corridor. Yellow indicates low density residential; red indicates commercial/industrial/transportation

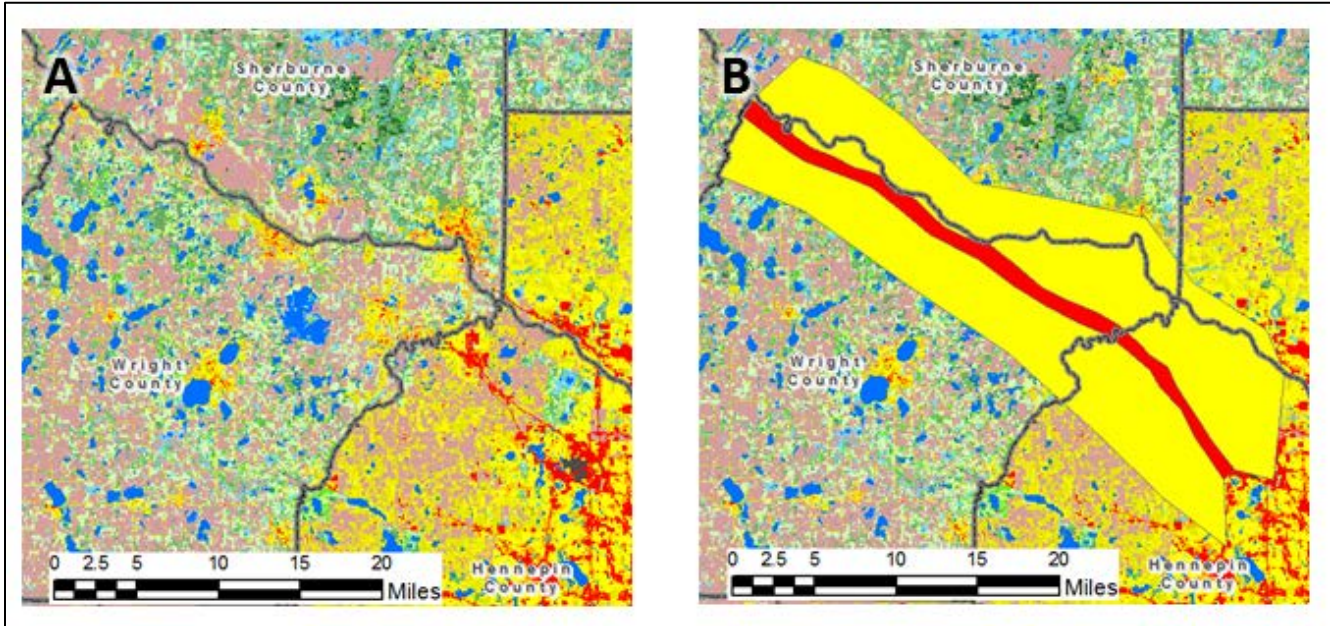


Figure 5.4-2 Attribute table for polygon shapefile of land use change

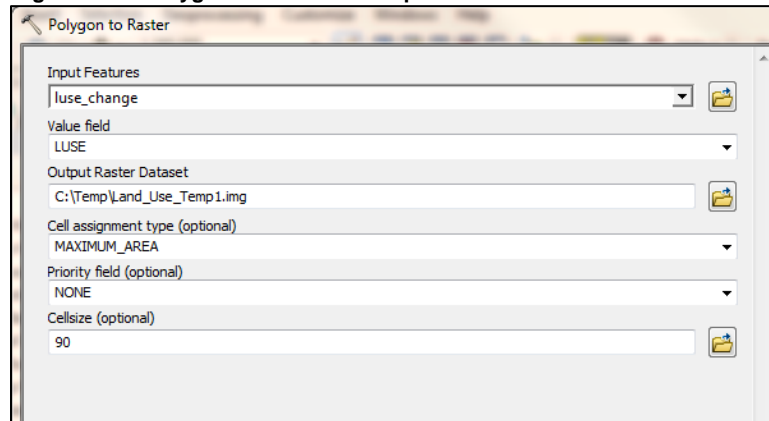
FID	Shape *	LUSE
0	Polygon	23
1	Polygon	21
2	Polygon	21

A shapefile of the land-use change, as shown above, is provided at *SWB_Tutorial\Misc\Change_Landuse\luse_change.shp*

The polygon coverage of the new land use needs to be converted to a raster with the same cell size and snapped to the existing land-use raster. **All input grids to SWB must be the exact same extent and cell size.**

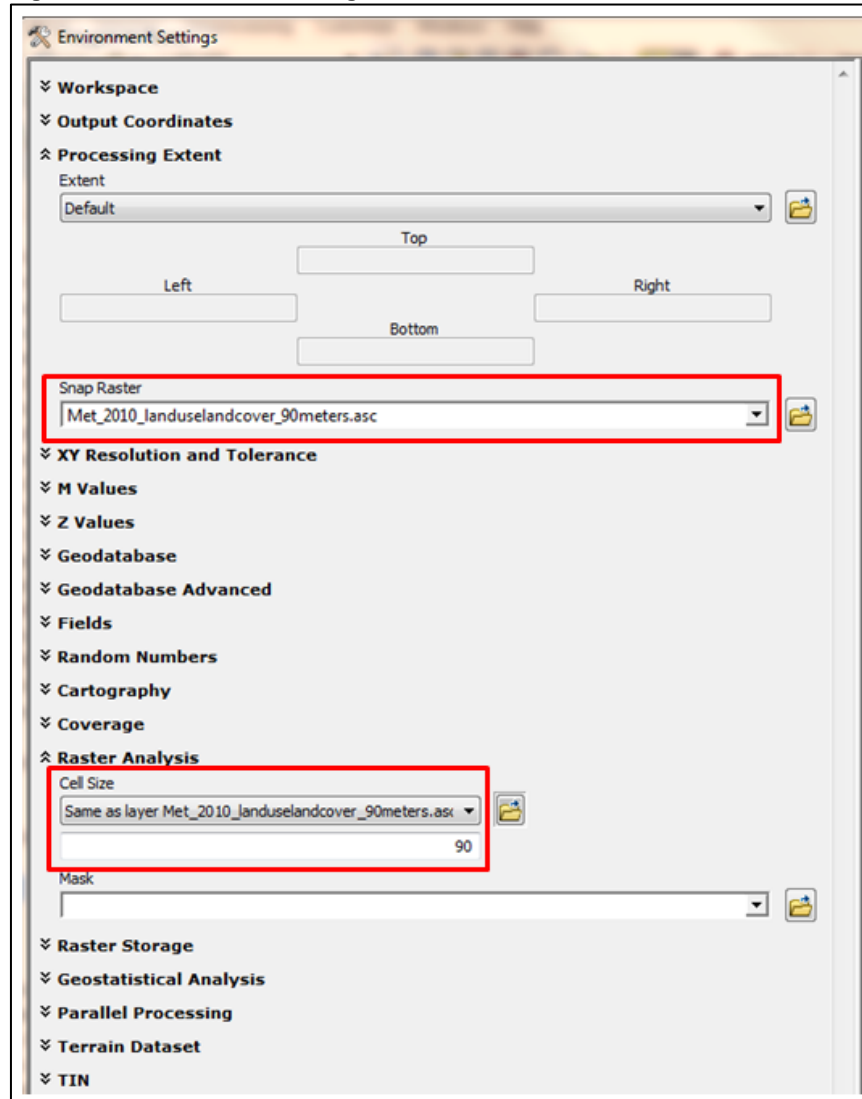
2. Add the existing land-use grid to ArcMap to use as reference for cell size and snapping. The existing land-use grid can be found at:
SWB_Tutorial\Misc\Change_Landuse\Met_2010_landuselandscape_90meters.asc
3. Use the **Polygon to Raster** tool to convert the polygon coverage of new land use to raster format. This raster will only be used temporarily and can be deleted later.

Figure 5.4-3 Polygon to Raster tool options



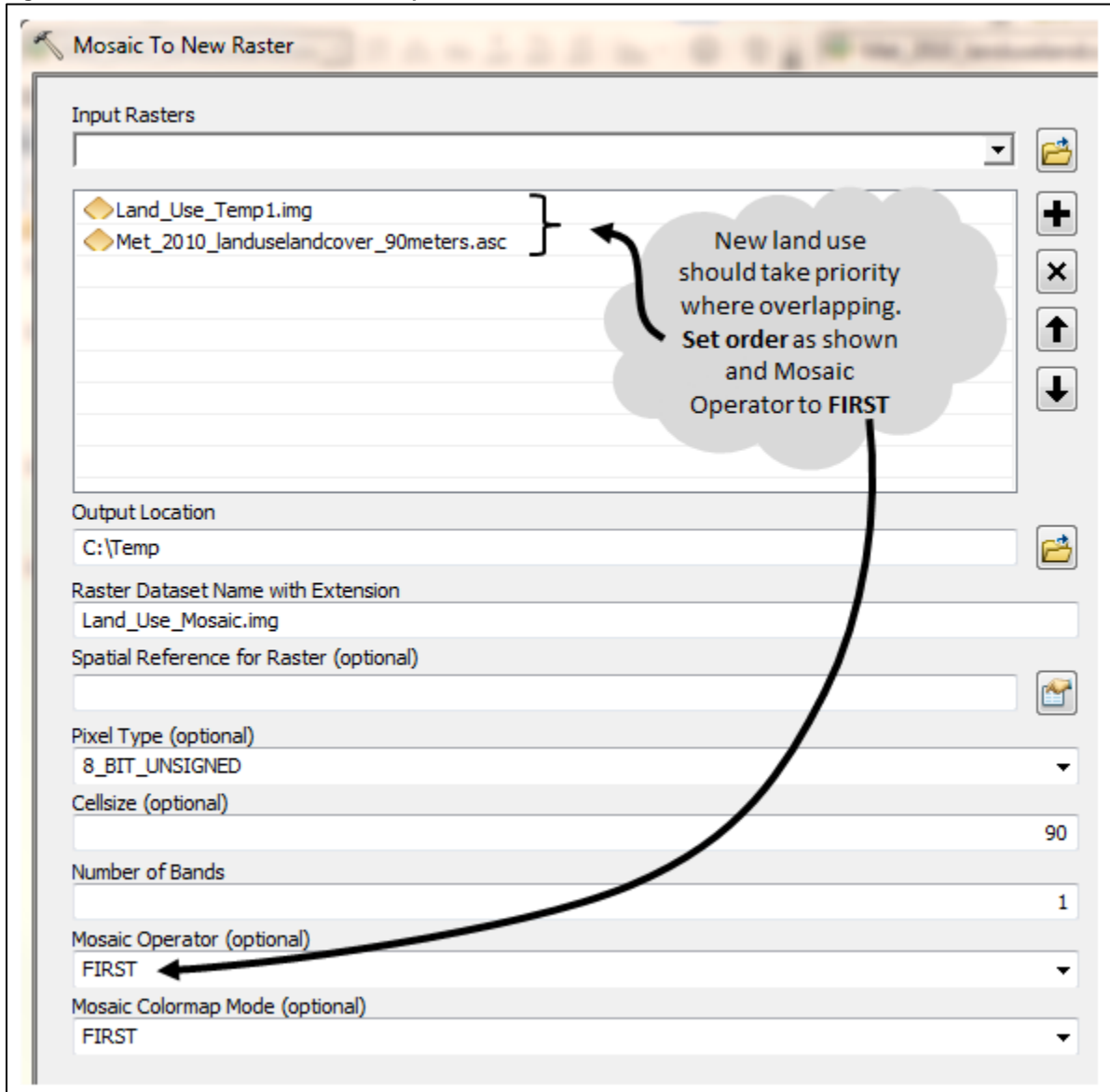
4. To properly set raster snapping and output cell-size, click on the **Environments...** button in the Polygon to Raster tool. Set the snap raster and cell size to the same as the existing raster dataset.

Figure 5.4-4 Environment settings



5. Mosaic the new land-use raster and the existing land-use raster into a new raster using the **Mosaic To New Raster tool**. Make sure that the new land use raster is set to be used for overlapping areas. As was done previously, set the snapping and cell size equal to the existing land use raster under the **Environments...** option button. This raster will also only be used temporarily and can be deleted at a later time.

Figure 5.4-5 Mosaic To New Raster tool options



A raster for the entire model domain is created with the new land use updated. However, areas such as lakes, wetlands, and existing commercial development have been over ridden. This is an unrealistic scenario for development along the corridor but is used here to illustrate the overall process.

6. Use the **Con** tool to fill in areas of existing open water (11), High Density Residential (22), Commercial/Industrial/Transport (23), and wetlands (92).

Figure 5.4-6 Con tool options

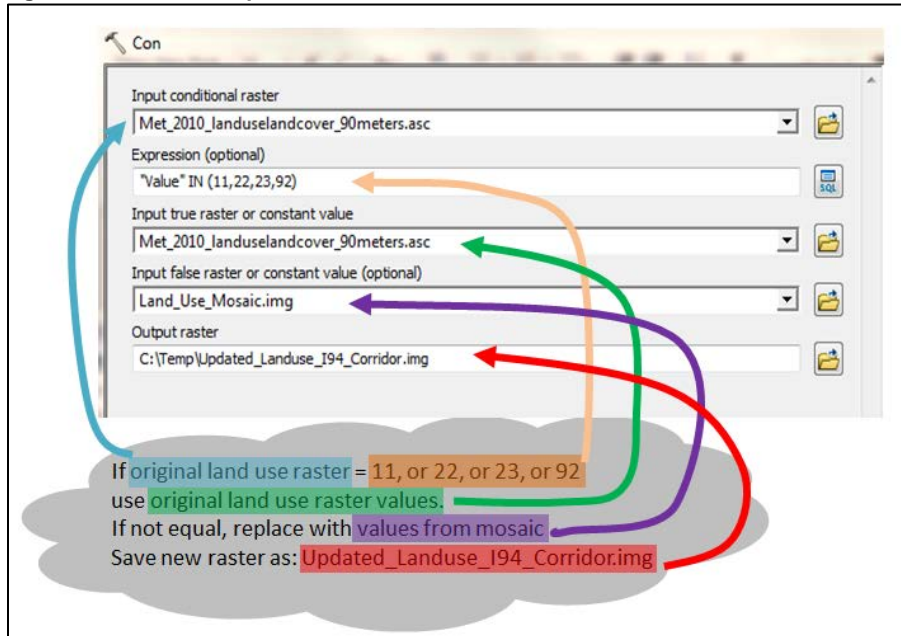
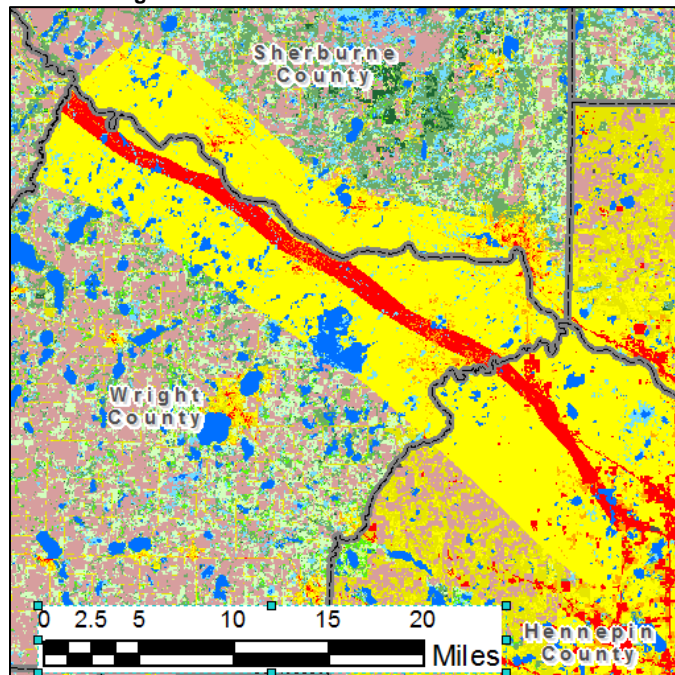
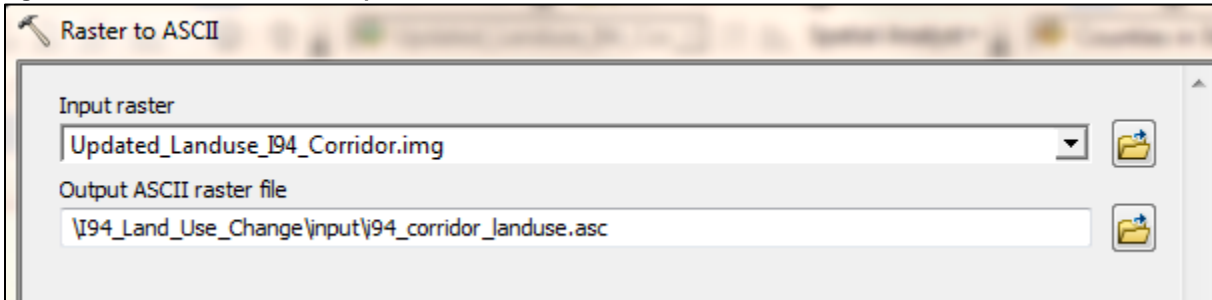


Figure 5.4-7 Updated land use raster grid with hypothetical land use change along I94 corridor. Only portion of grid where change made is shown



7. Export the new land-use raster to ASCII (.asc) format for use by SWB using the **Raster to ASCII tool**. This new raster should be saved to, or copied to, the `|I94_Land_Use_Change\input` folder of SWB project directory.

Figure 5.1-8 Raster to ASCII tool options



5.5 Modify SWB Control File

An example SWB control file is provided at `.\SWB_Project_Directory\SWB_Control.ctl`. Comments are provided within the control file describing the most commonly used commands. Changes to the control file needed for the simulations that are part of this tutorial are described in the following sections. The user is referred to the SWB manual for further description on the many commands available in the SWB control file.

5.5.1 Setting Climate Data in Control File

For SWB to run properly the SOLVE statement near the end of the control file needs to reference the correct climate time-series file.

1. Modify the **SOLVE statement** at the end of the control files as follows:
 - a. For `\Baseline\SWB_Control.ctl`:
SOLVE climate\1975.txt
 - b. For `\Climate_Change_2095\SWB_Control.ctl`:
SOLVE climate\2095.txt
 - c. For `\I94_Land_Use_Change\SWB_Control.ctl`:
SOLVE climate\1975.txt

If more than one year of simulation is desired, then there should be a separate climate time-series file for each year and multiple SOLVE commands should be present, one for each year.

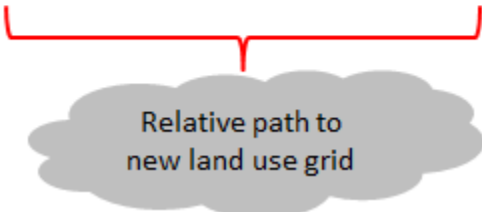
5.5.2 Setting New Land Use Grid

For the I94 corridor land-use change simulation the land-use grid referenced in the control file needs to be updated. The new land-use grid should have already been saved in ASCII (.asc) format in the `\I94_Land_Use_Change\input` directory as described in Section 5.4.

1. Update the **LAND_USE statement** in the `\I94_Land_Use_Change\SWB_Control.ctl` file as shown in Figure 5.5-1

Figure 5.5-1 Land use control statement in SWB control file

```
#*****  
#.....LAND USE/COVER CLASSIFICATION  
#*****  
# The model uses land use information, together with  
# the soil available water capacity information, to calculate surface  
# runoff and assign a maximum soil moisture holding capacity for each  
# grid cell.  
  
LAND_USE ARC_GRID input\i94_corridor_landuse.asc
```



Relative path to new land use grid

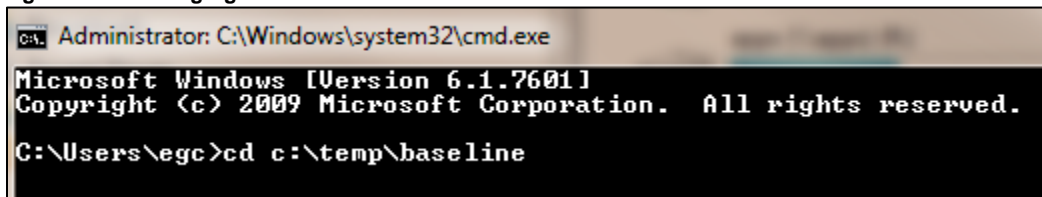
5.6 Running SWB

The SWB model must be run from the command window.

1. Open the command window. In Windows the command window can be accessed by typing **cmd.exe** in the search box under the Start Menu.
2. Once the command window is open the user must navigate to the models top level directory. For example, if the model is located at *C:\Temp\Baseline* the following command should be typed into the command window to navigate to that directory:

cd c:\temp\baseline

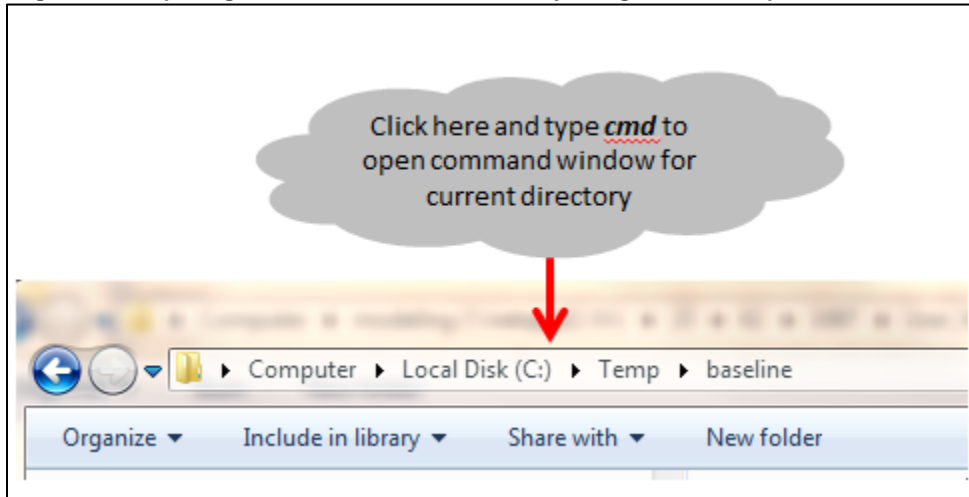
Figure 5.6-1 Changing directories in the command window



```
Administrator: C:\Windows\system32\cmd.exe  
Microsoft Windows [Version 6.1.7601]  
Copyright (c) 2009 Microsoft Corporation. All rights reserved.  
C:\Users\egc>cd c:\temp\baseline
```

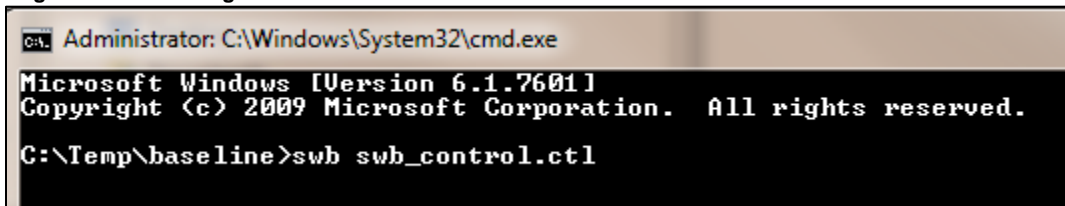
Alternatively, in some Windows environments if the folder of the top level directory is open in Windows explorer, **cmd** can be entered in the address bar and the command window will open already set to the correct directory.

Figure 5.2-2 Opening command window to directory using Windows explorer



3. Once the command window is active for the current directory SWB is run with the follow command:
SWB control_filename
4. Where *control_filename* is the name of the control file, including the extension suffix. It should also be noted that SWB.exe must be present in the top level directory.

Figure 5.6-3 Running SWB from the command window



5. Run SWB for each of the three simulations that have been set up; baseline, climate change 2095, and I94 land use change.

5.7 Comparing Results

Results from SWB output in raster grid format can be read into and compared using ArcMap. Alternative software such as Surfer, or Numerical Python (Numpy) could also be used, particularly if large batch processing is desired. However, use of alternative post-processing software is beyond the scope of this tutorial.

1. Open the annual average infiltration (recharge) .asc raster grids into ArcMap. These will be located in the `|output|annual` folder of the project directory. File names for these grids are automatically generated as `swb_RECHARGE_year.asc`. Where year is the year simulated as defined in the climate times-series input file. Units are inches/time period; in this case inches/year. If optional monthly grids were saved the units would be inches/month.
 - a. The year simulated for both the baseline and land-use change simulation was set at 1975; so, the annual output file names will be identical for these simulations. For convenience, the

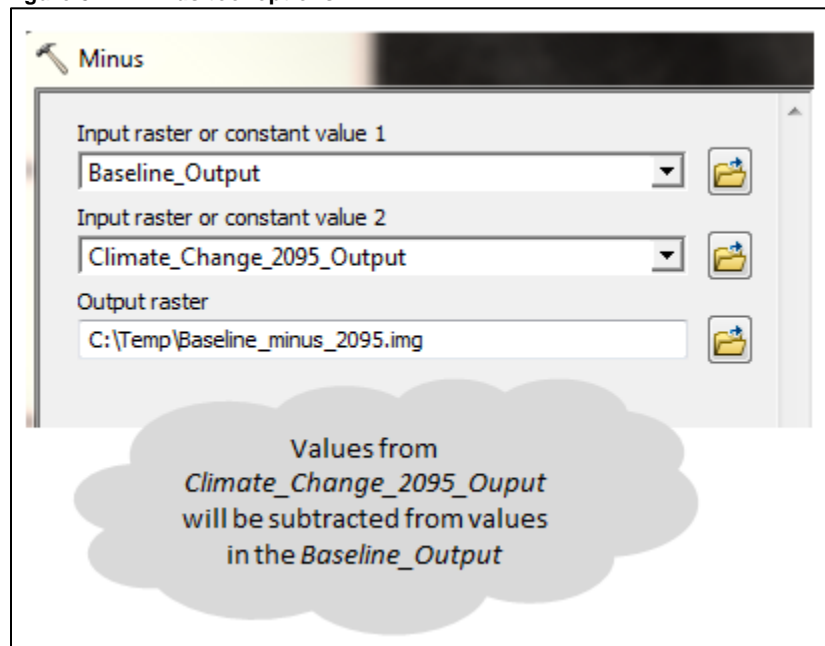
layer name within ArcMap may be changed to help keep track of the different output grids while post processing

- b. The output grids have coordinates representing UTM NAD 1983 Zone 15N. However, projection files are not generated by SWB and hence, ArcMap will not be aware of the projection. Projection files can be generated using the **Define Projection tool**, but is not absolutely necessary for comparing simulations.

When comparing different scenarios, the difference in the output (spatially) is typically of most interest.

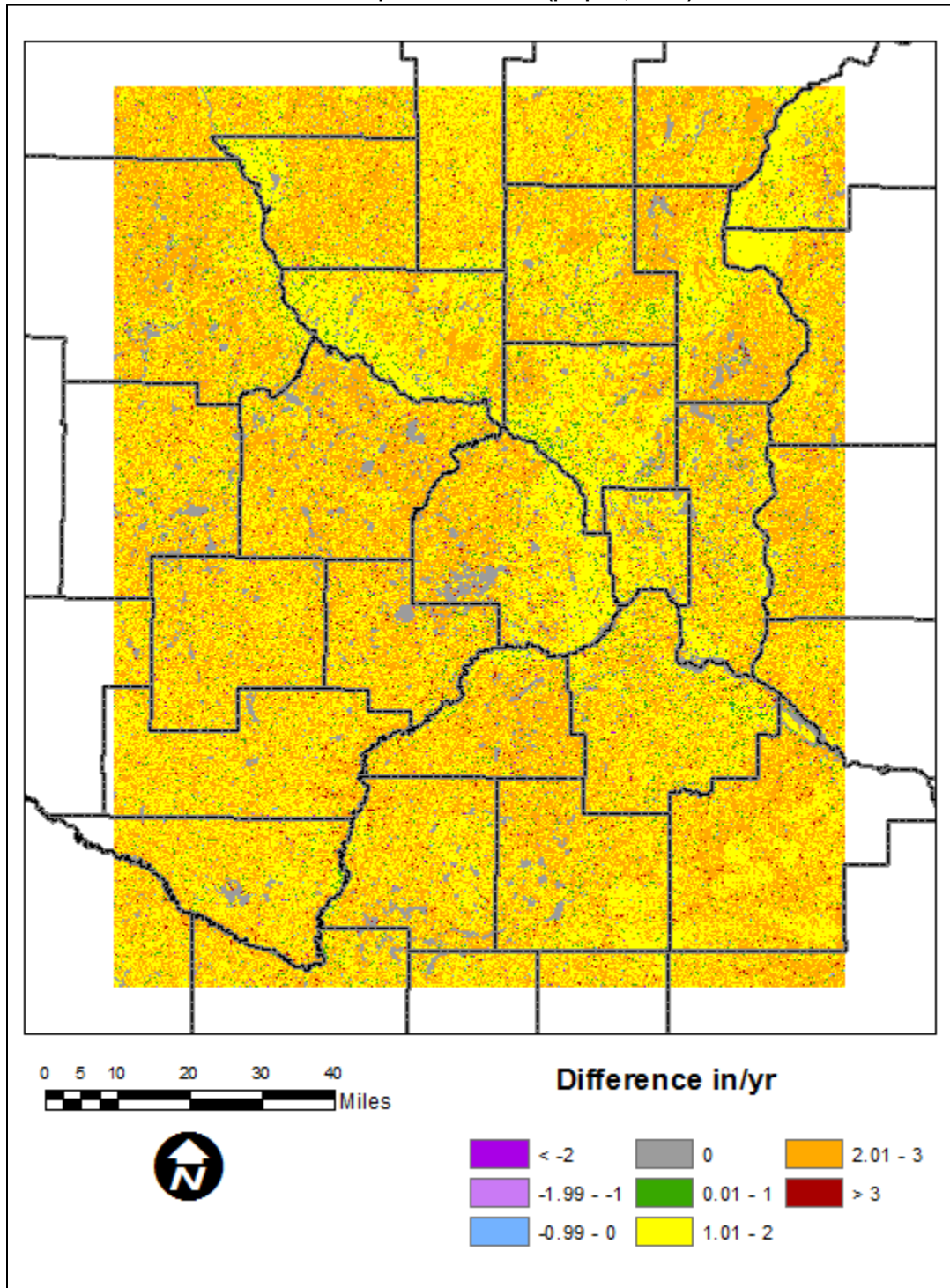
- 2. Use the **Minus tool** to compute the difference between the baseline simulation and the climate change simulation. Save the output to any convenient location. A spatial analyst license is necessary to use the Minus tool.

Figure 5.7-1 Minus tool options



The output grid from the Minus tool is the difference between the baseline simulation and the climate change simulation; where negative values represent an increase in infiltration (baseline < climate change) and positive values represent a decrease in infiltration (baseline > climate change).

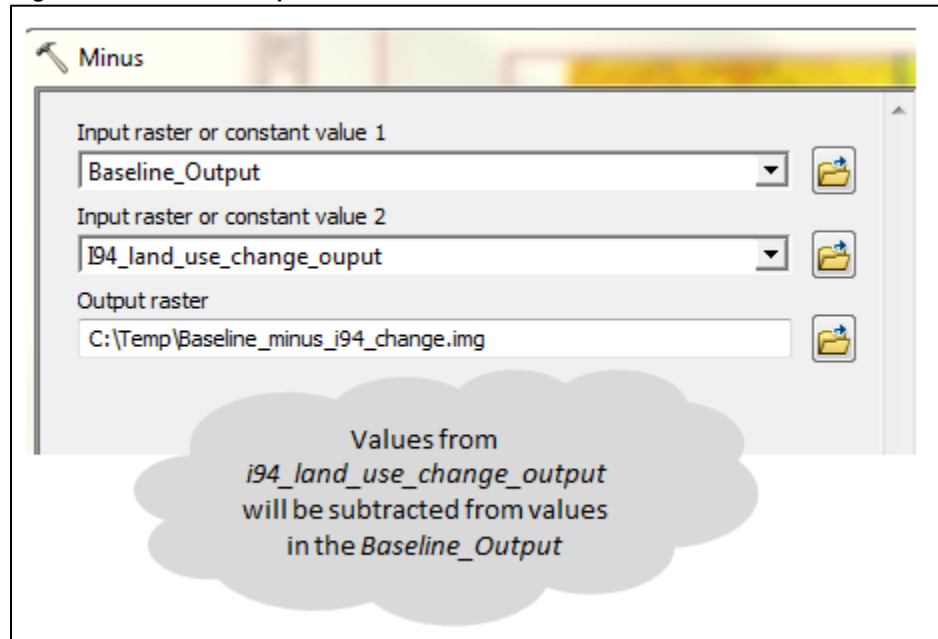
Figure 5.7-2 Difference between baseline simulation and 2095 climate change simulation. Positive values indicate a decrease in infiltration in 2095 compared to baseline (greens, browns, orange); negative values indicate an increase in infiltration compared to baseline (purples, blues).



As shown above, based on one simulation of climate change, infiltration will generally decrease in 2095 by 1 to 3 inches per year compared to baseline conditions. However, this accounts for only one stochastic climate time series. To accurately assess changes in infiltration many more realizations should be conducted.

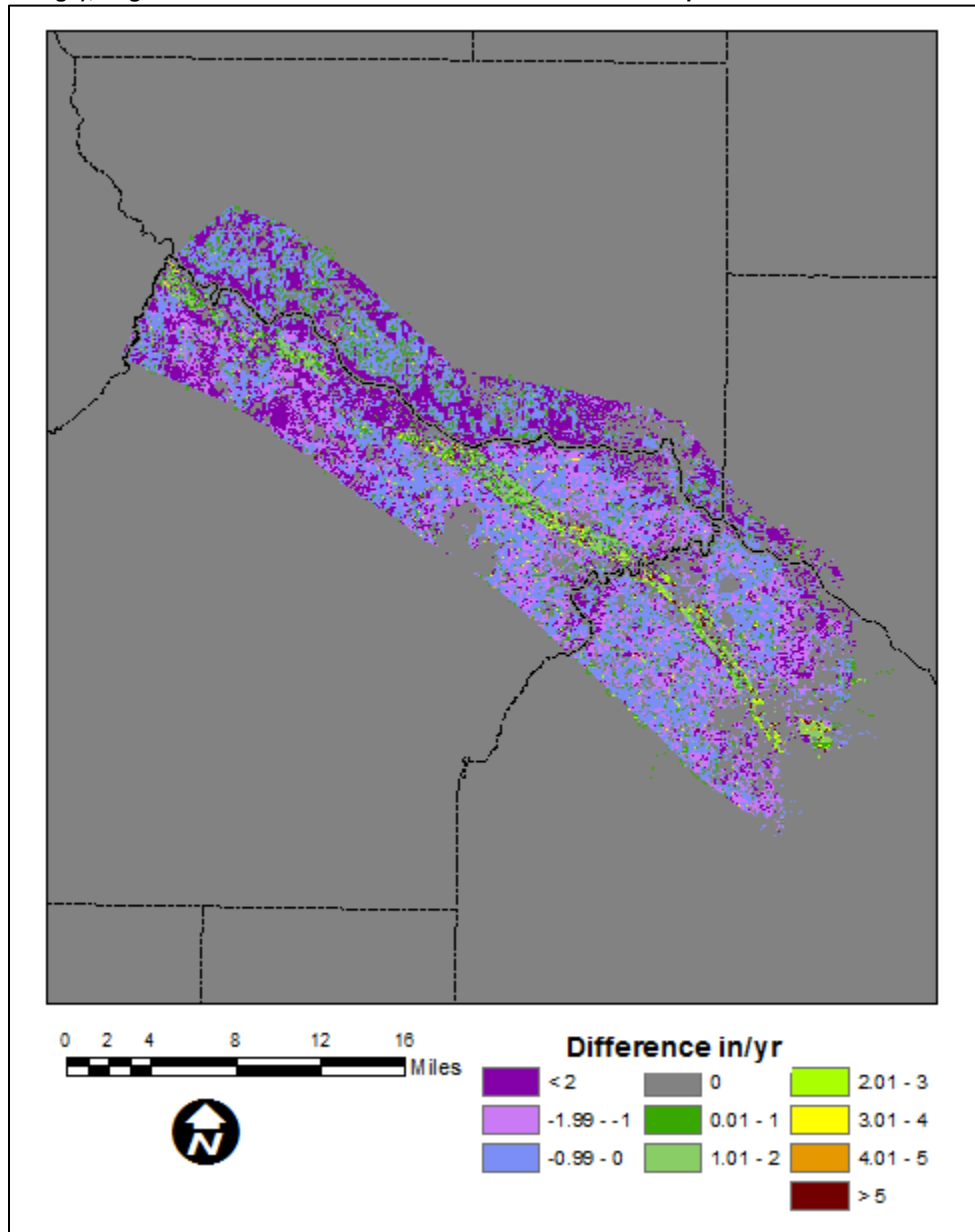
3. Use the **Minus tool** to compute the difference between the baseline simulation and the I94 land use change scenario. Save the output to any convenient location. A spatial analyst license is necessary to use the Minus tool.

Figure 5.7-3 Minus tool options



The output grid from the Minus tool is the difference between the baseline simulation and the I94 land use change simulation; where negative values represent an increase in infiltration (baseline < land use change) and positive values represent a decrease in infiltration (baseline > land use change).

Figure 5.7-4 Difference between baseline simulation and land-use change along I94 corridor simulation. Positive values indicate a decrease in infiltration in 2095 compared to baseline (greens, browns, orange); negative values indicate an increase in infiltration compared to baseline.



As can be seen above, based on one stochastic time series of climate, infiltration will generally increase between 0 to 2 inches per year in the area developed as low density residential. In the area developed as commercial/industrial/transportation, infiltration will generally decrease between 1 to 2 inches.

5.8 Using Gridded Climate Data

If gridded climate data are necessary for SWB input, the following apply:

1. Lines 50 and 71 in the example control file provided should be commented out by adding a “#” at the beginning of these lines. Lines 52 and 73 should be uncommented by removing the “#” at the beginning of the line. This signals SWB to use gridded climate data rather than single-station data.
2. Grids for daily precipitation, maximum temperature, and minimum temperature can be generated using any interpolation scheme. However, the grids should be saved in **ASCII (.asc) format**. Careful attention must be paid to the cell size and origin of the raster to be created. Each climate data grid must share a common datum, projection, grid-cell size, and grid extent with the land-use, flow-direction, soil-group, and soil-water-capacity grids. These settings can be controlled using the **Environment settings** in ArcGIS. Alternatively, an existing grid (such as land use) can be used to set the extent and snapping of any new climate data produced. The following apply:
 - Projection = NAD 1983 Zone 15N
 - Lower left corner easting = 373179.77926557
 - Lower left corner northing = 4885338.42926373
 - Cell Size = 90
 - Number of columns = 1810
 - Number of rows = 2232

ASCII grids have a common header. If generated correctly the header should read:

```
Ncols      1810
Nrows      2232
Xllcorner  373179.77926557
Yllcorner  4885338.4292637
Cellsize   90
NODATA_value -9999
```

Temperature grids should be **data type real** and in units **degrees Fahrenheit**. Precipitation grids should be **data type real** and in units of **inches/day**

3. Grids must be named using the following convention:

prefix_YYYY_MM_DD.asc

Where prefix is either *precip*, *tmax*, or *tmin* depending on the data, and **YYYY**, **MM**, and **DD** are the four-digit year, two-digit month, and two-digit day associated with the data.

4. All climate grids should be saved in one of the following folders: `.\climate\precip`, `.\climate\tmax`, or `.\climate\tmin` in the project directory.

Tutorial 6: Importing Files into Other Graphical User Interfaces

6.1 Introduction

Metro Model 3 was originally developed in Groundwater Vistas and then heavily modified, external of a Graphical User Interface (GUI), prior to bringing all data back into Groundwater Vistas. Standard MODFLOW files and formats are used. However, different GUIs offer varying levels of support for different MODFLOW packages. Below are generally the main methods used to import the raw MODFLOW files into another GUI for use.

6.2 Groundwater Modeling System (GMS)

Groundwater Modeling System (GMS) is a MODFLOW GUI developed by Aquaveo™. GMS is available for purchase from the Aquaveo website at www.aquaveo.com. At the time of writing, the processes described below fails because of a “bug” identified in the GMS software. However, the developers are aware of this bug and have indicated it will be fixed in a future release. Once fixed, the general process will remain the same. A GMS file with the model already imported has been provided by Aquaveo and is included with the tutorial files.

1. In GMS, select **File | Open**
2. Browse to the directory containing the Metro Model 3 model files and select the name (*.nam) file (File type drop-down may need to be changed to MODFLOW Name Files (*.mfn, *.nam)).
3. The MODFLOW Translator dialog will pop up. Select **OK**.
4. Once the translation is complete, all model files will have been imported

GMS does not yet support one option used in the UZF package for Metro Model 3. The NOSURFLEAK option must manually be added to the UZF package after creating the MODFLOW files and prior to running the model.

5. Select **MODFLOW | Run MODFLOW** to generate the model files. GMS may ask to save the model. It is OK to save at this step but note that the model should not be saved after step 8.
6. The model will run. Once running select **Abort** to stop the model. We need to change the UZF package manually before running to completion.
7. Navigate to the location where GMS saved the input files and open the *.uzf file in any text editor.
8. Add the keyword NOSURFLEAK to the first line of the *.uzf file and save the file. GMS does not follow standard MODFLOW input file formats as documented in the MODFLOW manual(s). However, adding this keyword does happen to follow the same convention as required in the USGS version of MODFLOW.

Figure 6.2-1 UZF package input file for GMS with NOSURFLEAK option added

```
NOSURFLEAK
3 1 0 0 0 -740 20 20 0 1.0
HDF5 1.0 -1 "MM3_Steady_State.h5" "UZF/12. IUZFBND" 1 0 139400
HDF5 1.0 -1 "MM3_Steady_State.h5" "UZF/16. VKS" 1 0 139400
HDF5 1.0 -1 "MM3_Steady_State.h5" "UZF/18. EPS" 1 0 139400
HDF5 1.0 -1 "MM3_Steady_State.h5" "UZF/20. THTS" 1 0 139400
1
HDF5 1.0 -1 "MM3_Steady_State.h5" "UZF/07. Property" 3 0 1 0 139400 0 1
```

9. Back in GMS, select **MODFLOW | Run MODFLOW**. This time if GMS asks whether changes should be saved, select **No**. If **Yes** were selected, the manual edit to the UZF package would be overwritten.
10. Let MODFLOW run to completion.

6.3 Model Muse

Model Muse is a free GUI for MODFLOW provided by the USGS. Model Muse can be downloaded from <http://water.usgs.gov/nrp/gwsoftware/ModelMuse/ModelMuse.html> . The current official release at the time of this writing (v. 2.19.1) does not support the importation of MODFLOW-NWT files directly (although it does support running MODFLOW-NWT). USGS staff working on Model Muse indicate that the next official release will be able to import MODFLOW-NWT files. A beta version is available from the author of Model Muse. This beta version (v. 2.19.1.34) was used in developing this tutorial. A Model Muse file was created using the process below and has been provided as part of the tutorial files.

1. In Model Muse, select **File | Import | MODFLOW-2005 or –NWT Model**.
2. In the Import MODFLOW-2005 Model dialog window enter 369875 for the X origin, 5088875 for the Y origin, and browse to the name of the raw Metro Model 3 MODFLOW files. The model origin as used by Model Muse is the upper left corner of the grid. Groundwater Vistas uses the lower left corner.

6.4 Visual MODFLOW

Visual MODFLOW is a GUI developed by Schlumberger Water Services. It is available for purchase from the Schlumberger website at <http://www.swstechnology.com>. At the time of writing Visual MODFLOW fails at importing Metro Model 3 files. All attempts to circumvent problems have been unsuccessful. The source of the problems has yet to be identified by Visual MODFLOW developers and there is no timeline for bug fixes.

References and Additional Resources

- Dripps, W.R., and Bradbury, K.R., 2007, A simple daily soil-water balance model for estimating the spatial and temporal distribution of ground-water recharge in temperate humid areas: *Hydrogeology Journal*, v. 15, p. 433-444.
- Environmental Simulations, Inc., 2011. Guide to using Groundwater Vistas, Version 6, Environmental Simulations Inc.
- Harbaugh, A.W., 2005, MODFLOW-2005, The U.S. Geological Survey modular ground-water model—the Ground-Water Flow Process: U.S. Geological Survey Techniques and Methods 6-A16, variously p.
- Jones P.G., Thornton P.K. and Heinke J., 2011. Generating characteristic daily weather data using downscaled climate model data from the IPCC Fourth Assessment. <https://hc.box.net/shared/f2gk053td8>
- Konikow, L.F., Hornberger, G.Z., Halford, K.J., and Hanson, R.T., 2009, Revised multi-node well (MNW2) package for MODFLOW ground-water flow model: U.S. Geological Survey Techniques and Methods 6-A30, 67 p.
- Metropolitan Council, 2012. Using the soil water balance model (SWB) to estimate recharge for the Twin Cities Metropolitan Area Groundwater Model Version 3. Prepared by Barr Engineering. Metropolitan Council: Saint Paul, MN.
- Metropolitan Council, 2014. Twin Cities Metropolitan Area Regional Groundwater Flow Model, Version 3.0. Prepared by Barr Engineering. Metropolitan Council: Saint Paul, MN.
- Niswonger, R.G., Prudic, D.E., and Regan, R.S., 2006. Documentation of the unsaturated-Zone Flow (UZFI) Package for modeling unsaturated flow between the land surface and the water table with MODFLOW-2005: U.S. Geological Survey Techniques and Methods 6-A19, 62p.
- Niswonger, R.G., Panday, S., Ibaraki, M., 2011. MODFLOW-NWT, A Newton formulation for MODFLOW-2005: U.S. Geological Survey Techniques and Methods 6-A37, 44 p.
- Westenbroek, S.M., Kelson, V.A., Dripps, W.R., Hunt, R.J., and Bradbury, K.R., 2010, SWB—A modified Thornthwaite-Mather Soil Water Balance code for estimating ground-water recharge: U.S. Geological Survey Techniques and Methods 6-A31, 60 p. The Soil Water Balance code is available for from the USGS at: <http://pubs.usgs.gov/tm/tm6-a31/3>