A Groundwater Flow Model for the Little
Plover River Basin in Wisconsin's Central Sands

User's Manual

Kenneth R. Bradbury
Anna C. Fehling
Michael J. Parsen

UW Extension
University of Wisconsin-Extension

Contents

NOTE: The Wisconsin Geological and Natural History Survey obtains its Groundwater Vistas licenses from ESI; however, several other providers also distribute the software.

We do not endorse one product over others.

Introduction

This document provides basic instructions and recommendations for running the steady-state and transient versions of the Little Plover River (LPR) groundwater flow model, including use of the Groundwater Vistas (Vistas) user interface. The model is a state-of-the-art groundwater flow and optimization model for the Little Plover River basin in Portage County, Wisconsin. The steady-state model was designed to represent average groundwater conditions given a recent time period; the transient model represents one calendar year, with calibration focusing on 2014, but can be modified to simulated longer time periods or different years.

This publication is intended to assist technical users in running the model; for a comprehensive discussion about the development, construction, and calibration of the model, refer to the modeling report, A Groundwater Flow Model for the Little Plover River Basin in Wisconsin's Central Sands (Bradbury and others, 2017).

The model can be run directly using native MODFLOW files, but most users will need a graphical interface, such as Groundwater Vistas, to view and edit model input and results. A version of the Groundwater Vistas model including all necessary files, as well as the native MODFLOW files, are available for download from the UW-Box folder dedicated to the LPR model: https://uwmadison.app.box.com/v/little-plover-river.

This report references Groundwater Vistas as a method to modify datasets and visualize model output. Vistas is a Windows-based graphical user interface using a series of embedded windows and tabs. Accessing various menus requires that the user select (mouse click) through a menu series, and in this manual we express these series in sequence. For example, Model > MODFLOW > Packages means that the user would click on the Model tab, then on the MODFLOW tab, then on the Packages tab to bring up the menu of interest.

This model was last modified using version 6.93 build 34 of Vistas, so users should update to at least this version of Vistas before attempting to run the model.

Additional resources

Groundwater Vistas (software)

The Help menu at the top of the screen offers extensive documentation and assistance troubleshooting problems.

Groundwater Vistas: Online user manual
groundwatermodels.com/
Groundwater_Vistas.php

Environmental Simulations Incorporated (developer of Groundwater Vistas)
groundwatermodels.com

Technical support, webinars, and seminars for Groundwater Vistas.

MODFLOW: Online guide (USGS)
water.usgs.gov/nrp/gwsoftware/
modflow2000/MFDOC

MODFLOW: Report (USGS)
pubs.usgs.gov/tm/2005/tm6A16

Background information and settings

Before you begin, you may find it helpful to familiarize yourself with the model layers used, as well as key settings and a brief comparison of the two conditions—steady state and transient—for which the model was calibrated.

Model layers

The LPR model has three layers, which represent the following specific hydrostratigraphic units:

Layer	Unit simulated
1	Upper unlithified sand and gravel (glacial deposits and alluvial materials)
2	Lower unlithified sand and gravel (glacial deposits and alluvial materials)
3	Cambrian sandstone

The sand and gravel aquifer is represented by two layers of approximately equal thickness to improve simulation of vertical gradients and flow near surface water features. All model layers are treated as continuous across the model domain. In places where specific formations, such as the Cambrian sandstone, are absent (eroded away), the model layer thickness is set to 1 ft and the layer made effectively transparent by assigning hydraulic conductivity values equal to those of the layer above. The crystalline rock below the sandstone forms the base of the model.

Model units, orientation, and GIS options

The model units are in feet and the time unit is days, which can be viewed here: Model > MODFLOW > Package Options, under the Basic tab. The real-world spatial units or map units (called "site coordinates" in Vistas) are in meters. This is important to remember when importing or exporting basemaps, well locations, or other features into or out of Vistas. The coordinates of imported features need to be in meters and coordinates of exported features will be output in meters if the output is a shapefile. By contrast, if you are exporting ASCII grid files, such as a Surfer grid, these will be output in model coordinates (feet), which do not reflect the selected offset

coordinate system. Attributes, like hydraulic conductivity or recharge, are reported in ft/day.

To orient the model properly with map files when exporting to GIS, the GIS options (under Edit > GIS Options…) should be set as shown in the screenshot.

The projection file is a GIS projection file (.prj) that specifies the correct GIS coordinates. You can leave this blank if you prefer to project the output shapefiles later or browse for a desired .prj file that you are currently using. We commonly use NAD 83 HARN Wisconsin TM in meters.
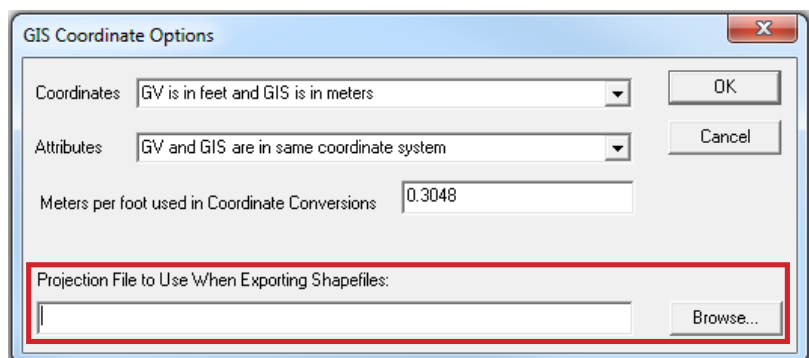
The model has a uniform cell spacing of 100 feet on a side. There is no grid rotation.

Basic information about the model grid (number of rows, columns, layers, cells), grid spacings, and model coordinates are summarized in the Model Summary window (Model > Model Information).

Steady-state and transient models

The LPR model was calibrated to both steady-state and transient conditions. The steady-state model represents average equilibrium conditions where heads or flows do not change over time, while the transient model is capable of accounting for changes in head or flows over time. The steady-state model was developed first and is generally calibrated to average hydrogeologic conditions between 2010 and 2014; the anomalously dry conditions of 2012 were not included. The steady-state model served as the starting point for transient calibration, which simulates the 2014 calendar year. Calibration targets used for history matching included heads, stream flows, and lake stages.

While some sections of this guide are relevant to both model conditions, the text generally refers to the steady-state version. For information that is exclusively about the transient model, see the section titled "Transient Simulations."

Orient the model with map files when exporting to GIS.
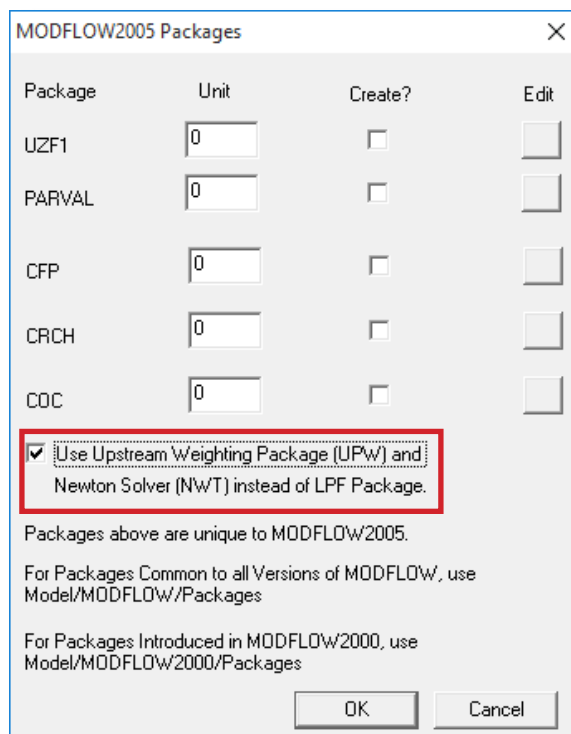
Technical setup

Newton Solver

This model uses the USGS Newton formulation of MODFLOW-2005 called MODFLOW-NWT. This is a sophisticated solver that is intended for problems involving drying and rewetting of cells. In Vistas, this solver is included in Model > MODFLOW2005 > Packages. While MODFLOW-NWT is already selected in the LPR model, it is important to understand where to activate the Newton Solver. To implement it in Vistas, the checkbox next to "Use Upstream Weighting Package (UPW) and Newton Solver (NWT) instead of LPF Package" must be checked in the Model > MODFLOW2005 > Packages window.

The solver parameters are set under the Model > MODFLOW2005 > OPTIONS > NWT General tab. However, these options have been optimized for best model performance and in general should not be changed. Note also that the solver unit number, which MODFLOW uses to keep track of input and output files, is still set under the Model > MODFLOW > Packages tab even though the solver method specified on that tab is ignored.

Directories

Be sure to properly set up the working directory prior to running the model on a new machine (Model > Paths to Models). The working directory is the folder where all output files will be stored. The executable files are also set up using the Paths to Models box; this is described in a separate section ("Running the Model"). You may also choose to change the root file name for the output files under Model > MODFLOW > Packages.

Check box to activate Newton Solver

Map files

The steady-state model is loaded with three map files already in Vistas .map format. These files must be kept in a separate directory and loaded when Vistas starts up. Several other map files have been included that you may elect to import.

The map files include the list shown below.

Two of the files we have provided are in bitmap (.bmp) format. Only one bitmap file may be loaded at a time. To load a bitmap file, select File > Map > Bitmap. You will be prompted to enter the map coordinates. Note that although the bitmap is in meters, the coordinates must be entered in feet. We have provided the coordinates in the file "BMPconverter.xls." Copy and paste from this spread-sheet into the Vistas window. Additional instructions ("How to Add a Bitmap as a Basemap") are also included with the map files.

To load additional maps, go to File > Map and select the format of the map you want to import. This allows you to browse for a map and add it to Vistas. All maps that are loaded are viewable in Plot > Map Overlays…. From this window you can select to draw, not draw, or delete individual maps. You can also change the thickness of the map lines. The display of maps can also be controlled by going to Plot > What to Display… and selecting or deselecting the box Display Base Maps. For a map to be viewable it must be set to Draw in the Plot > Map Overlays window.

Map files included with the model

File name	Description
LPR_HUC_watershed.map	LPR basin 12-digit HUC (hydrologic unit code) watershed
Port_Co_maj_roads.map	major roads (Portage Co.)
Port_Co_min_roads.map	minor roads (Portage Co.)
Del Monte app areas.map	Del Monte irrigation application areas
Port_co_lakes.map	lakes (Portage Co.)
Port_Co_streams.map	streams (Portage Co.)
LPR_ortho_bitmap_base.bmp	orthophoto (model area)
LPR_topo_bitmap_base.bmp	topographic map (model area)

Running the model

There are two common ways of running the LPR groundwater model. The first is using the native executable file (MODFLOW-NWT.exe) from USGS. This method is faster, but requires some initial setup and does not display onscreen results of how the model is running. In general, we recommend this method as it uses the "official" and most up-to-date USGS solver. The second method is using the MFNWTWIN32.dll executable supplied with Vistas. This option takes longer to run but is easy to set up because it is the default method in Vistas. The two methods are described in more detail in the following sections. Two additional methods that are less commonly used are described in "Running the Transient Model."
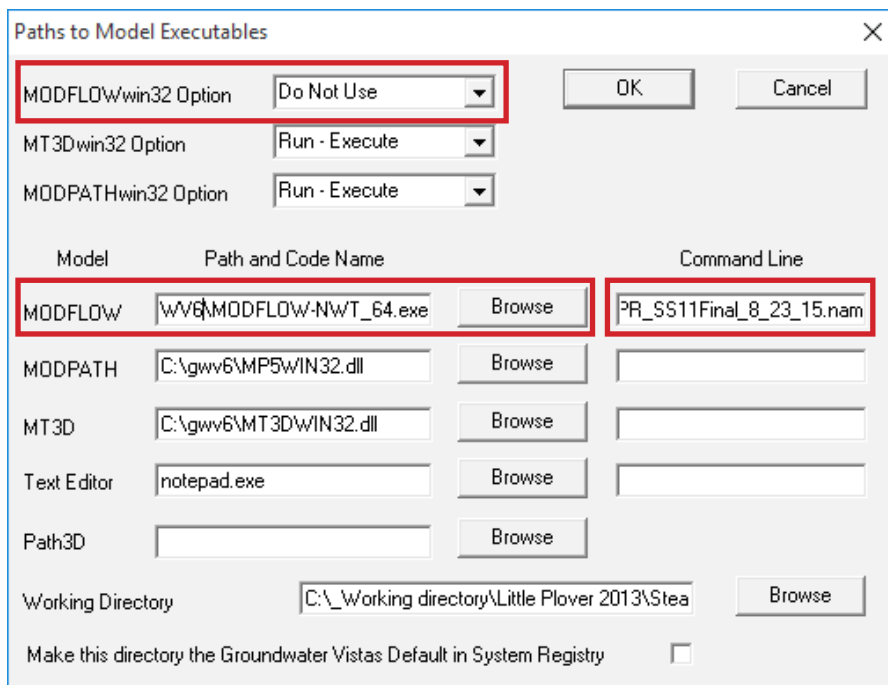
NOTE: **You must use version 1.0.9 of the USGS NWT Solver (MODFLOW-NWT).** Newer versions may not work. The program is included with the model files.

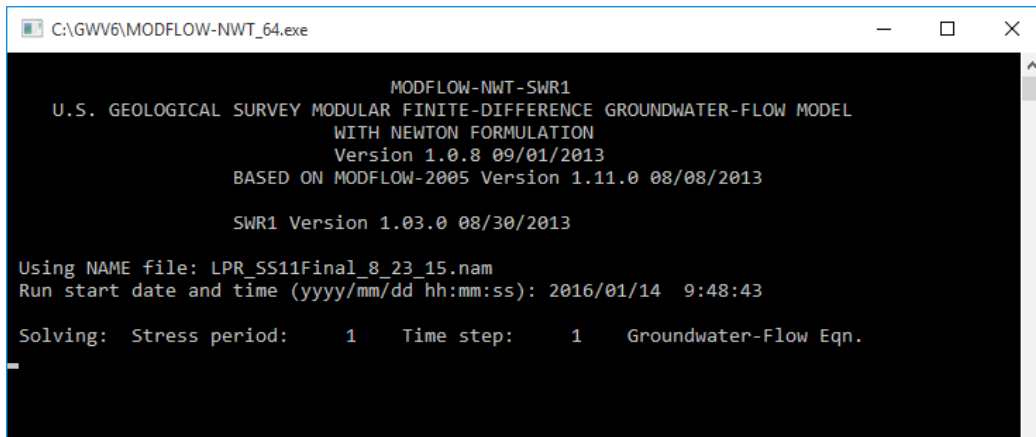## By running the USGS executable (.exe) file in Vistas

The model can be run in Vistas using the executable file from USGS (MODFLOW-NWT.exe or MODFLOW-NWT_64.exe). This method requires that you change two settings in Vistas and understand a bit more about how the executable file reads and writes all the other files using the name (.nam) file. It displays only limited feedback during the run. However, you can check on the progress of a model run by opening the list (.lst) file. For more details, see instructions under "Monitoring a Transient Run."

To run the model in this way, you first need to download the appropriate .exe file from the MODFLOW folder on https://uwmadison.app.box.com/v/little-plover-river and save to your computer.

In Vistas, create datasets by clicking on Model > MODFLOW > Create Datasets. This will create a number of files including a name (.nam) file which are saved in the working directory (see "Directories"). The name file is a list of all the input and output files needed to run the model. Open the directory to confirm the name file is there. Then, in Model > Paths to Models (see screenshot), set the MODFLOWwin32 Option to "Do Not Use," and set the MODFLOW path to wherever the MODFLOW-NWT_64.exe file is saved. In the Command Line box, enter the name file.

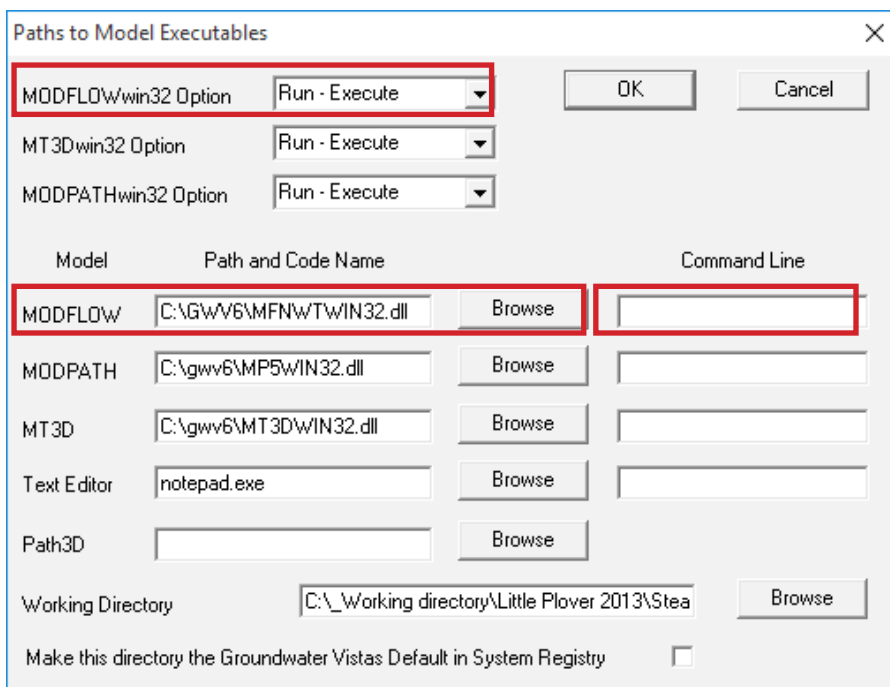Settings used to run the USGS executable file

DOS window showing the model running

Once this has been set up, you can run the model (Model > MODFLOW > Run Model). A DOS window will pop up to show you the model has started running (see screenshot).

Note that the following message will immediately pop up in Vistas: "MODFLOWwin32 is finished! Process the results?" This message can be misleading because the model is running using the executable file, not MODFLOWwin32. You can either wait for the model to run before clicking "yes", or you can click "no" to make the box disappear. This won't interrupt the run and will let you perform other tasks on the computer while the model runs in the background.

By running the .dll file in Vistas

This method is easiest to implement, as it is the default method in Vistas. It is visually more satisfying because it yields a continuous readout of what the model is doing and the error criteria as they change. This comes at a cost though, as the computer must use overhead to write to the screen, so running the steady-state model takes a few minutes longer than the other method, and can essentially tie up your computer, making multitasking very slow. In addition, it does not use the official USGS solver so may give slightly different results or be less computationally stable than the USGS solver.

To use the Vistas .dll file, go to Model > Paths to Models (see screenshot) and set the MODFLOWwin32 option to Run-Execute. Next, set the Path and Code Name to MFNWTWIN32.dll. You will need to browse to find the .dll file in the folder where you installed Vistas on your computer. You do not need to enter anything in the command line.

From here you can create datasets and run the model through the Model > MODFLOW menu.

Settings used to run the .dll file in Vistas

Viewing model results

After running the model, you will select what results to import (see screenshot). Check the box next to the files you wish to import. If you are looking at flows, you must import the cell-by-cell flow file.
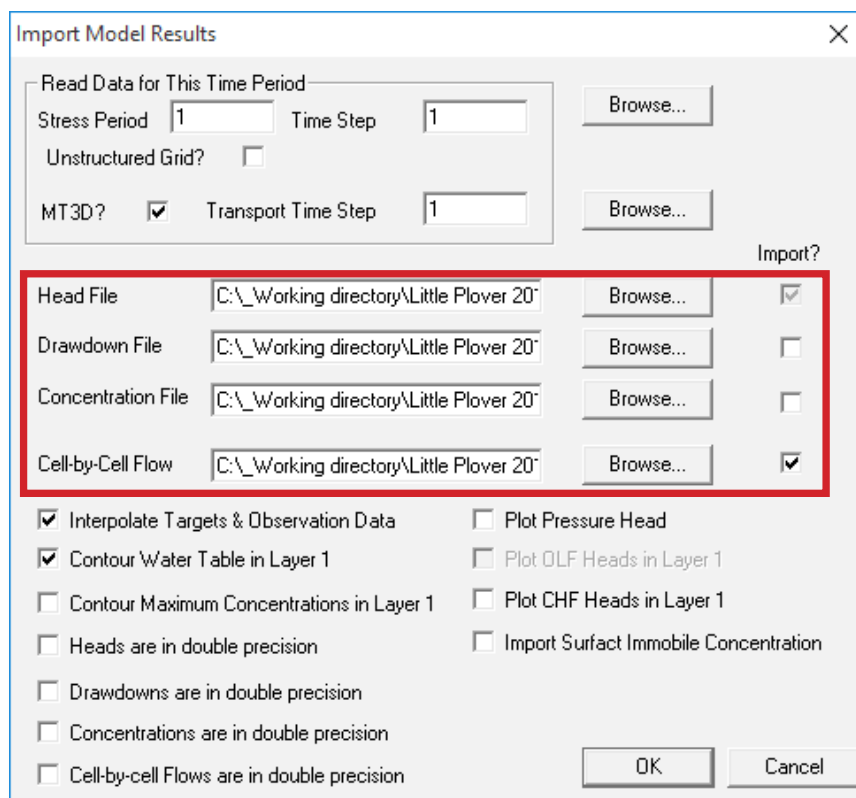
Run time

Run time depends on the processing speed of the computer used, other processes running on the computer, and the complexity of the MODFLOW problem. The steady-state LPR model generally runs in under 3 minutes on a typical 64-bit machine. Longer run times will occur if model stresses are substantially changed by adding or removing wells.

Model output

Running the model produces a number of ASCII output files that contain useful information. The main binary files (.hds, .ddn., and .cbb) are used to import heads, draw-down, and cell-by-cell flow into Vistas. The list file (.lst) is another useful file, which can be opened using a text editor to view the model budget at the bottom of the file.

Additional output files can be specified by editing the output control file. To edit the output control file, go to Model > MODFLOW > Package Options…, on the Output Control tab, check the box next to "Use Custom Output Control," and specify the number of custom steps. By doing this, the default settings are deactivated and you will need to set up the output control under Model > MODFLOW > Custom Output Control. Custom output control is most useful during transient runs and is addressed in the "Transient Simulations" section of this manual.

Select results to import for viewing

Viewing and editing model features

Streamflow routing (SFR2 Package)

All internal streams and several drainage ditches are routed using the Streamflow Routing (SFR2) Package in MODFLOW, which accounts for base flow accumulated from stream leakage. An important feature of the SFR2 Package is that streams can lose water and even go dry if the groundwater head drops below the stream base or the head in the stream drops to the stream base.

> **NOTE:** Routed streams are identified by numbered segments and reaches. The number and order of these reaches is very important and should not be changed.

Note that modeled flow in streams represents baseflow only, and runoff from storm events is not simulated. However, there is little topographic relief and high connectivity between surface water and groundwater in the Little Plover River basin. Baseflow was estimated to be approximately 80% of total flow in the Little Plover River (Bradbury and others, 2017).

A discussion on how to view SFR2 output follows in "Understanding Streamflow Reporting (SFR2 Package)."

Users may wish to manually add discharges at discrete locations to represent human-made discharges to the LPR. This can be manually entered in the Model > MODFLOW2000 > SFR Segment Data… window in the Flow column.
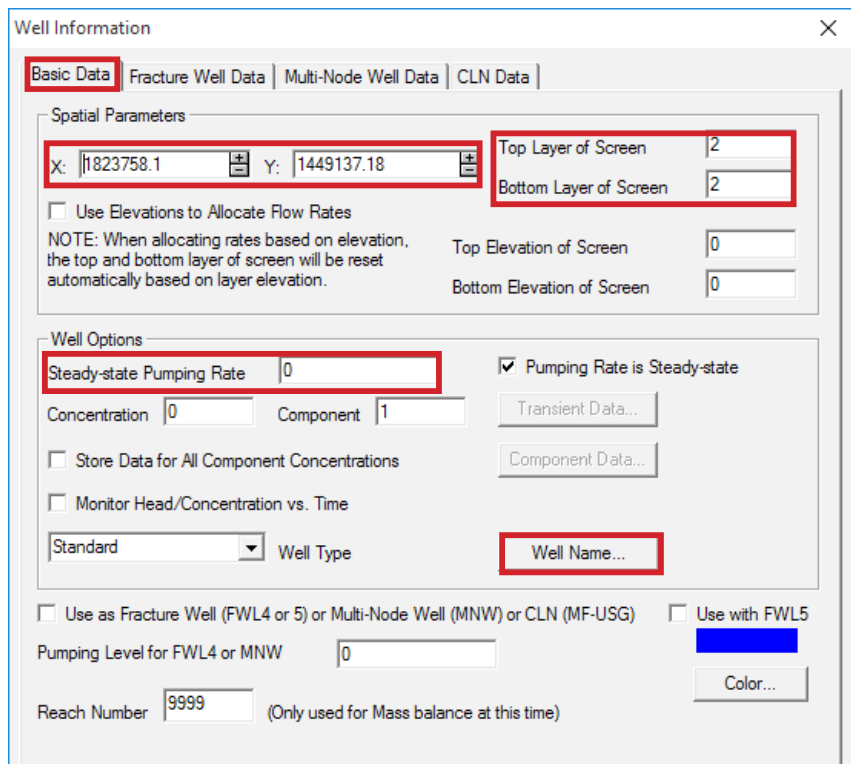
Other surface water features

Lakes are simulated as high-hydraulic conductivity features (1x10^6 ft/day) in layer 1. You can view these features by selecting Props > Hydraulic Conductivity. When represented in this way, lakes are essentially a reflection of the water table. One wetland, the Jordan Swamp, is represented by the MODFLOW Drain Package (to view, see BCs > Drain). Drains are head-dependent flux boundaries that can only remove groundwater from the system.

Wells

This model uses the standard MODFLOW Well Package (WEL). To view a well in Vistas, the Analytic Element (AE) button must be selected and you need to be in the model layer(s) from corresponding to the open, or uncased, interval of the well. Click on the well symbol to view its properties.

Adding a new well

To add a new well, you will need to activate the Add Well tool. To do this, zoom in to where you would like to add the well, and click AE > Well. Then click the mouse button where you would like to insert the new well; a pop-up window will appear for you to add the necessary information about the new well. As shown in the screenshot,

Adding a new well—Basic Data tab settings

set the exact coordinates of the new well location (X, Y), the model layers corresponding to the open interval of the pumping well (Top Layer of Screen, Bottom Layer of Screen), and the steady-state pumping rate. A negative pumping rate represents a withdrawal from the ground-water system, while a positive pumping rate represents an injection. Pumping rates must be entered in cubic feet per day, and so a unit conversion is typically necessary. In this tab you can also name the well. Keep the default settings for all other fields and tabs. For adding a transient well, in which pumping rates change with time, see the "Transient Simulation" section of this manual.

Editing an existing well

To edit an existing well, such as increasing the pumping rate, click on the well to open the Well Information window and make the changes to the Basic Data tab.

Removing an existing well

To remove existing wells, they first need to be selected. Either click on a single well directly, or select multiple wells using Edit > Select By or Edit > Select All. Then press the delete key to remove the wells. You can confirm that wells were removed by checking the total number of wells listed on the model information window (Model > Model Information). If you are interested in deleting all wells, for example to simulate predevelopment conditions, you can simply select all wells (Edit > Select All > Well) and press delete.

Recharge

Groundwater recharge varies spatially across the land-scape in this model. Initial values of recharge were taken from a modified soil-water-balance model (see Bradbury and others, 2017), and were adjusted regionally during model calibration. In this model, recharge represents deep drainage from the soil zone, or the water that actually reaches the water table. Evapotranspiration and applied irrigation are explicitly accounted for in the recharge values and are not directly input into MODFLOW. Refer to the model report for a more detailed discussion of recharge and how it can be applied to land use analysis.
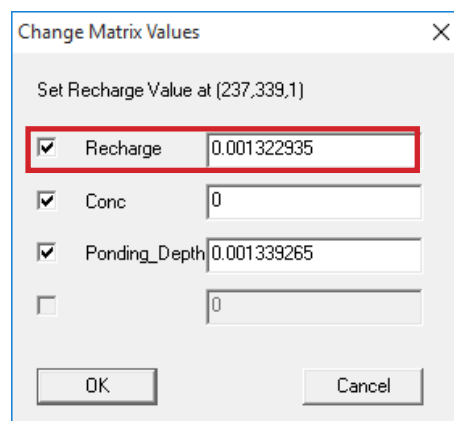
Recharge can be viewed by selecting Props > Recharge. The lower left corner displays recharge in units of feet/day.

Users may wish to modify the recharge to simulate the steady-state conditions following land development or a change in climate. However, recharge should be modified with caution due to the complexity of the initial value distribution and the unknown consequences of future climate change (such as increased pumping during a drought).
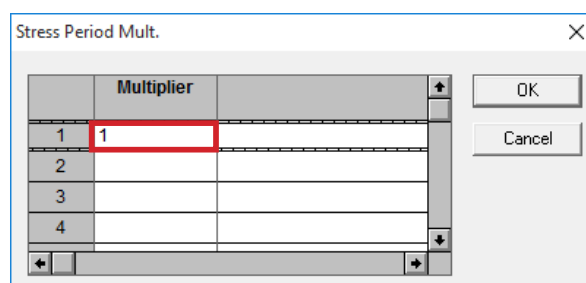
To directly edit recharge in a specific cell or area, select Props > Set Value or Zone. The screenshot below shows how to set recharge at a single cell. To set a given value of recharge over a larger area the user can choose Window to highlight a rectangular area or Digitize Polygon to highlight an irregular area and then set the recharge in the selected area to a single uniform value using the Change Matrix Values tool below.

Model-wide changes to recharge may be simulated using multipliers. To do so, first confirm that Recharge is selected from the Props menu. Then, select Props > Property Values > Recharge Multipliers. Enter the desired multiplier.

In the Little Plover River model it is likely that users might want to add or subtract recharge to a pre-existing condition (such as by increasing irrigation to a field) rather than setting nodes to a common value. Vistas has a tool called the matrix editor to do this. To access it, select Props > Property Values > Matrix Editor. This brings up a matrix spreadsheet that allows you to perform math calculations on specific blocks of cells. The example on the following page shows how the matrix editor would be set up to add 0.001 ft/day of recharge to a range of cells from row 525, col 363 to row 552, col 389 in layer 1. Note that the same method works for other gridded properties as well. The area of change must be rectangular to use this method. For irregular or circular areas it will usually be necessary to set up the recharge array outside of Vistas using GIS or other software and then import the new array.

Setting recharge for individual cells

Use multipliers to make global changes to recharge

Hydraulic conductivity

Hydraulic conductivity varies spatially across the landscape. To view hydraulic conductivity values, select Props > Hydraulic Conductivity. Values are displayed in units of ft/day in the lower left corner.
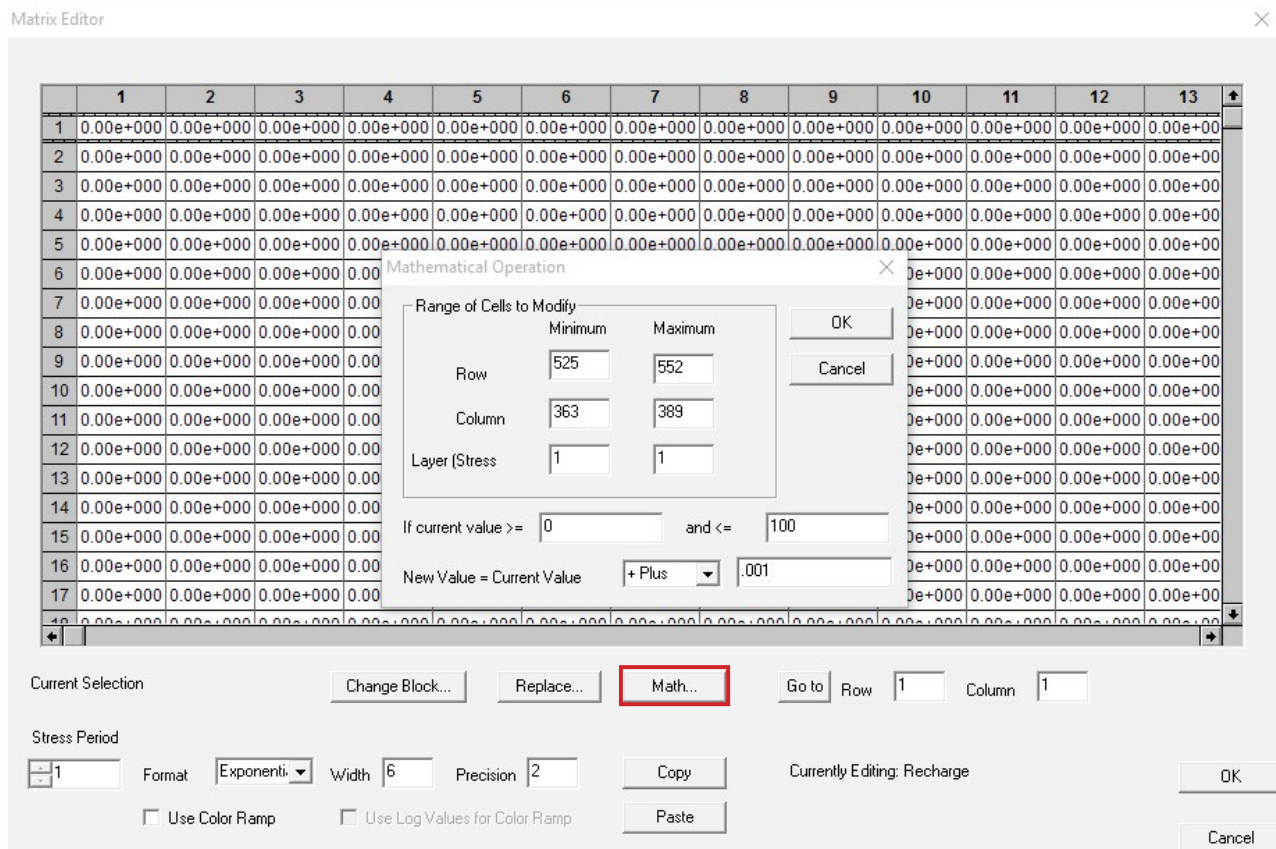
Understanding streamflow reporting (SFR2 Package)

The Streamflow Routing (SFR2) Package is a powerful modeling package that calculates the amount of water entering (from groundwater) and leaving (to groundwater) each stream cell, as well as the cumulative flow rate in the stream at each stream cell. This section describes how to use SFR2 output to determine the mass balance of water entering and leaving each stream cell in the model. Understanding how groundwater and surface water interact is an important component of many water resource evaluations.

Following a run, once model results (including the cell-by-cell flow .cbb file) have been imported, Vistas can be queried to show the simulated streamflow at any stream point. To do this, you must turn on Stream Boundary

TIP: Ordinarily, Vistas reports streamflow in the base model units (feet and days in the case of the Little Plover model). This means that streamflows and flux are reported as cubic feet per day as large numbers such as $8.64x10^4$ cubic ft/day, which might not be intuitively meaningful. However, Vistas has an option of setting a multiplier to change the displayed value. Select Plot > What to Display and near the bottom of the pop-up menu there is a box titled Multiplier on Status Bar Fluxes. The reported flux values will be multiplied by this number. So, for example, to change the reported units to cubic feet per second you would enter 1/86,400 sec/day, or $1.157x10^{-5}$. Then the streamflow will be reported in the more common units of cubic feet per second (cfs).

Condition (BCs > Stream) or specify Stream in the drop-down Boundary Condition box. Next, hover the cursor over any stream node (you must be in the model layer containing the SFR2 stream node, typically layer 1 or 2). The lower

Matrix editor in Vistas

left corner of the screen will report the following information about the node:

- **s (segment)** = a long segment of the stream (we'd normally call this a reach)
- **r (reach)** = an individual node along that segment
- **H (head)** = the hydraulic head in the stream
- **F (flux)** = flux into or out of that particular reach (node). Note that a negative value means flux out of the groundwater system and into the stream node; a positive value means flux into the groundwater system and out of the stream node.
- **Flow** = accumulated baseflow in the stream. Flow equals the sum of all individual node fluxes from the first stream node (upstream) up to and including the node of interest. This number should always be positive, or zero if the stream is dry.

Based on the screenshot at right, the streamflow report should be understood as follows:

- s (segment) = 715
- r (reach) = 4
- H (head) = 851.63 feet above mean sea level
- F (flux) = -1.397×10^4 cubic ft/day (again, negative flux means water is moving out of the groundwater system and into the stream)
- Flow = 3.642×10^5 cubic ft/day (accumulated baseflow of stream up to, and including this node)
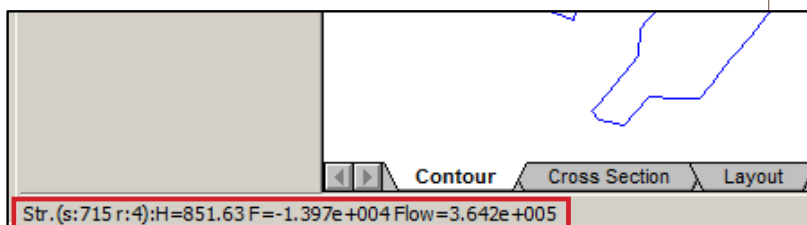
To better understand this, go to the most upstream segment of any stream and hover the cursor over that node. For this node, flux (F) should equal the negative of flow as reported by Vistas. Some headwater SFR2 nodes may report flux and flow equal to zero, indicating that they are simulated as dry.

To view all SFR2 cells at once go to the Boundary Conditions (BCs) drop-down window (select Streams), press Options… and check the box Display All BCs When Viewing Layer 1. This will allow you to view all SFR2 cells as long as you are in model layer 1. As you drop down to lower layers, only the SFR2 cells present in that layer will be visible. To see which SFR2 cells are present only in layer 1, uncheck the box (see screenshot).
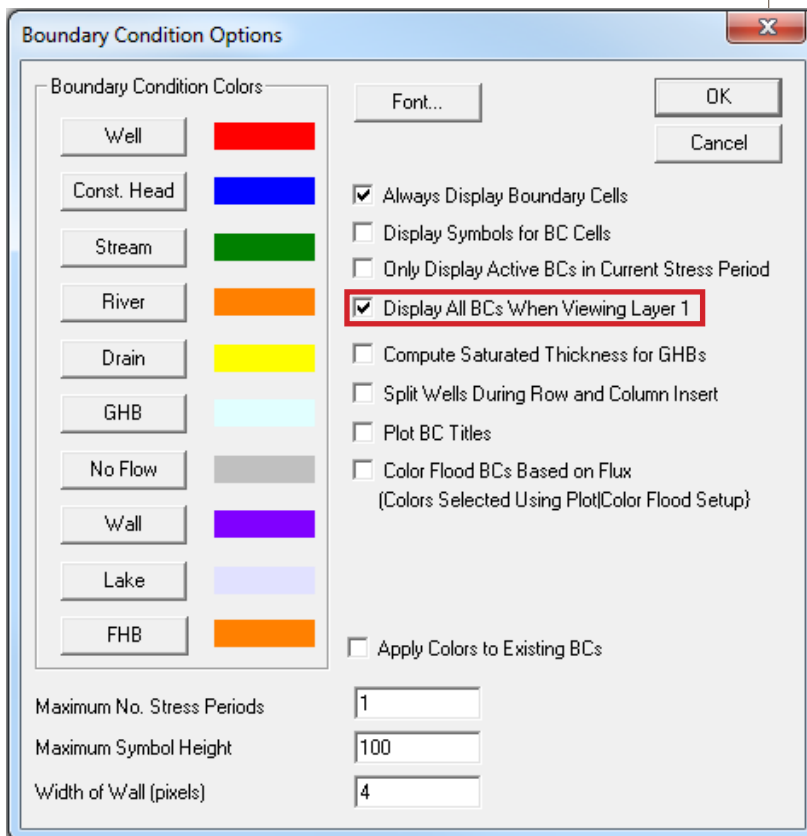
To manipulate and study streamflow results, it is best to export the streamflow data as

a shapefile from the Boundary Conditions (BCs) drop-down window (BCs > Export > Shapefile) and work with the data in GIS. When exporting a shapefile of the streamflow results, name the shapefile accordingly and select the shape type. Polygon (model cells) will export polygons representing the model cells while the type Point (nodes) will export a point feature class of nodes located at the center of each stream cell. Either way the same streamflow attributes (s, r, H, F, and flow) will be exported for all stream cells.

NOTE: We have developed an ArcGIS macro for easily comparing streamflow results. See ArcMap SFR2 Baseflow Comparison macro.

Sample streamflow report

Uncheck box to view SFR2 cells present in layer 1 only

Particle tracking using MODPATH

Particle tracking is a powerful way to help visualize the direction of groundwater flow and evaluate the impact of new or existing wells, simulate flow away from a landfill, or towards a subdivision. Particles can be simulated in both forward and reverse particle tracking mode, allowing the user to investigate groundwater flow both up-gradient and down-gradient from a specific feature of interest. This chapter provides a basic overview for how to perform particle tracking using MODPATH in Groundwater Vistas.

Currently the model is compatible with MODPATH version 5 (which is included with Vistas). Use the MP5WIN32.dll executable in Model > Paths to Models.

Add particles to your model by going to the Analytic Element (AE) drop-down pane (AE > Particle) and selecting an appropriate shape for inserting particles. Particles can
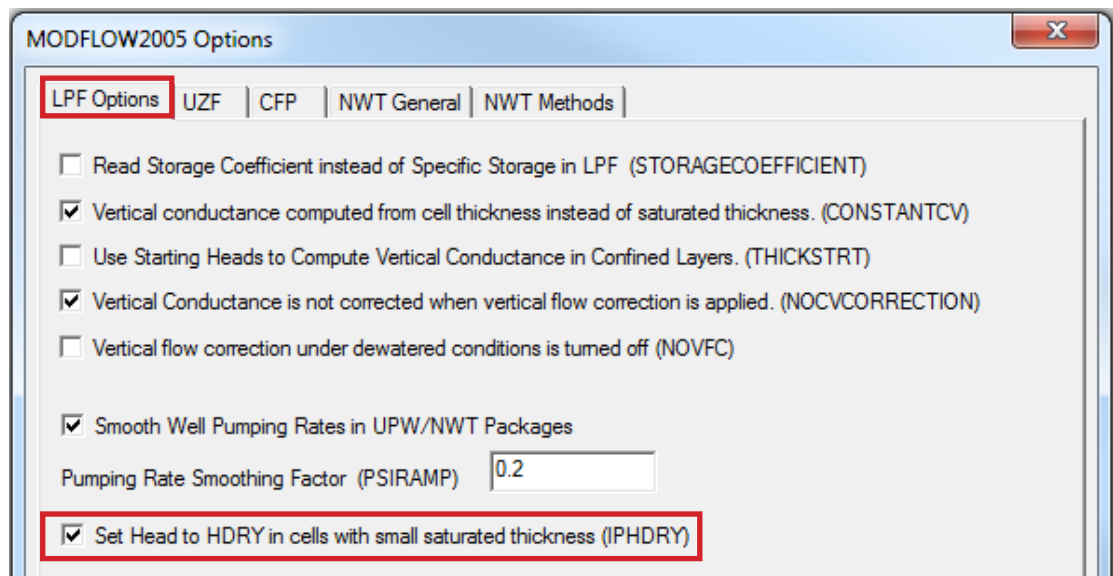
be inserted as single particles, lines, circles, window or digitized polygon features. Circles of particles work best for delineating capture zones of wells, while lines, windows and digitized shapes of particles work well for delineating flow paths to or from larger features, such as lakes and streams.

The following example demonstrates how to perform a capture zone (area of contribution) analysis for a well by delineating a 100-year time-of-travel area.

Start by setting up the particle options under Model > MODPATH > Particle Options…, setting the Direction of Tracking to Reverse, checking the box Limit Particle Travel

NOTE: MODPATH v.5 will crash if the program encounters dry cells. To avoid this bug, *before* running MODFLOW, go to Model > MODFLOW2005 > Options, select the LPF Options tab, and put a check in the box next to "Set Head to HDRY in cells with small saturated thickness (IPHDRY)" (see screenshot).

TIP: If the box highlighted below is checked, **the resulting heads cannot be used to calculate drawdown** between two model runs because many wells will start out dry. To get around this, see the section "Calculating Drawdown."

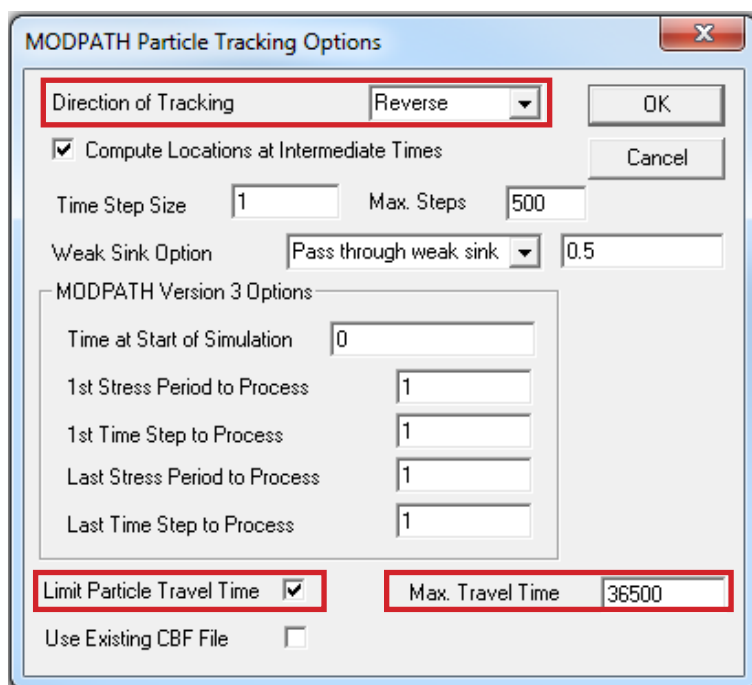**Check box to keep dry cells from causing MODPATH to crash**

Time and entering Max. Travel Time in days (100 years x 365 days = 36,500 days). If you do not limit the particle travel time in this window, the particle traces will be infinite. We recommend leaving all the other check boxes and settings as shown in the screenshot.

If you are viewing particle traces in Vistas only (not exporting as a shapefile), you may wish to not select a maximum travel time since this allows you to view any desired particle travel time by adjusting Maximum
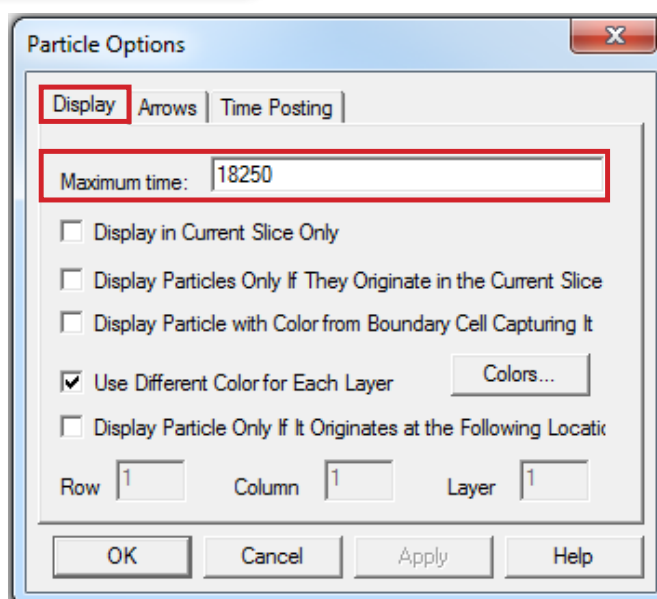
Time in Plot > Particles > Options under the Display tab. A number of other particle display options are available in this window, including color coding particles based on model layer or adding arrows and time postings along the particle travel path.

Once you have set the particle options in Model > MODPATH > Particle Options…, the next step is to insert circles of particles for the well of interest. Under the Analytic Element (AE) drop-down window, select AE > Particle > Circle. When you click on Circle, the window will disappear and you can hover over the model space and click a spot where you would like to add a circle of particles. If you continue to hold down the button, you can adjust the diameter of the particle circle. When you release the mouse button, a pop-up window will appear next to the particle circle where you can enter circle trace information (see screenshot on the next page).

Settings for particle tracking

Displayed maximum travel time can be adjusted here if viewing particle traces in Vistas only (not exporting)
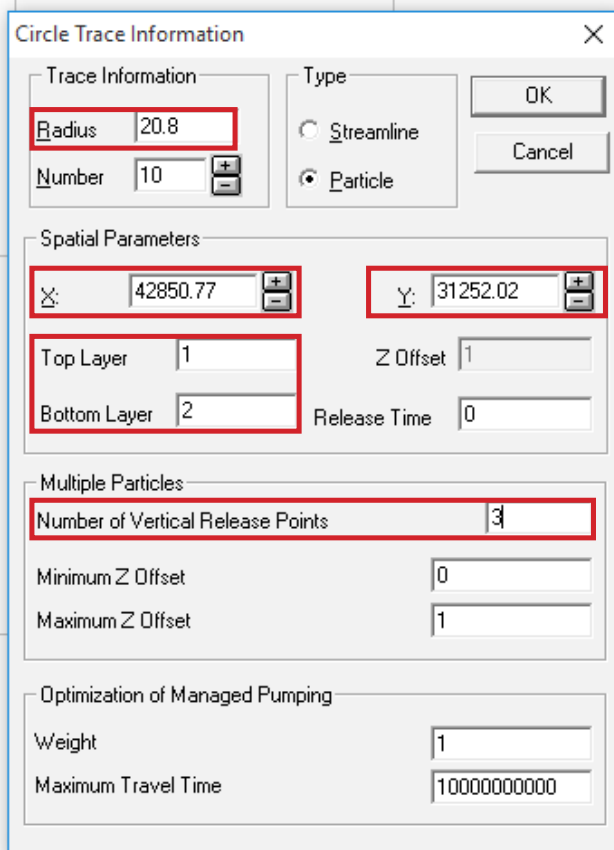
The initial location and size of the particle circle is not critical since these settings can be adjusted in the pop-up window. You can also double-click on the particle circle (just as you would with a well) to reopen the window. As for wells, to select a particle circle, the Analytic Element (AE) tab must be selected and you need to be in the same model layer containing the particles.

> **NOTE:** The effective radius for the near-field of the Little Plover River model is calculated as 100 ft (model grid interval) ÷ 4.81 = 20.8 ft.

Place the particles as close as possible to the center of the model cell. To aid in this, we recommend making the model grid cells visible (go to Plot > What to Display, and check Display Finite-Difference Grid). While the well you may be interested in is not necessarily located in the exact center of the model cell (as in the screenshot below), the center of your particle circle needs to be in the center to correctly perform a particle trace for the well. This is because the numerical model solution is solved for the model node, which is located at the exact center of the model cell. One simple way to do this is to
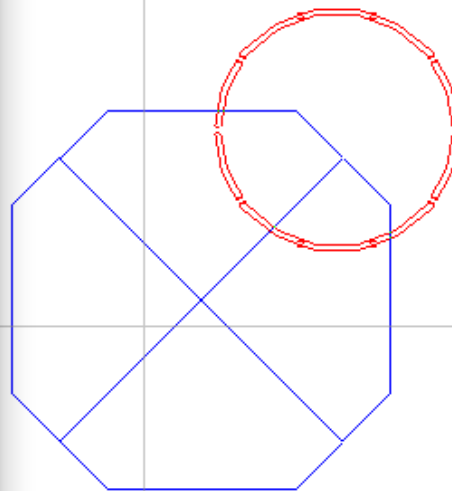
first make the diameter of your particle circle equal to the cell size (set the circle radius to 50 ft since the cell size equals 100 x 100 ft), and then manually adjust the spatial parameters (coordinates X and Y) by trial and error until the circle exactly fits in the middle of the model cell. Once the particle circle is centered, the radius can be changed back to 20.8 ft. The radius of the particle circle should be 20.8-ft based on the size of the model cells (100 x 100 ft) and the principles of converging flow near a pumped well node (Prickett and Lonnquist, 1971).

When doing reverse particle tracking to delineate capture zones for wells, we recommend using three circles of 10 particles in each open interval layer of the well. In other words, if the well is open between model layers 1 and 2 you would place three circles of 10 particles in both layers (a total of 6 circles and 60 particles). You will also need to specify the open interval of the well (top and bottom layers) and the Number of Vertical Release Points to 3. For

> **TIP:** Specify the layer for adding particles directly from the pop-up window. This is often easier than clicking down to each model layer and adding circles for a particular model cell.

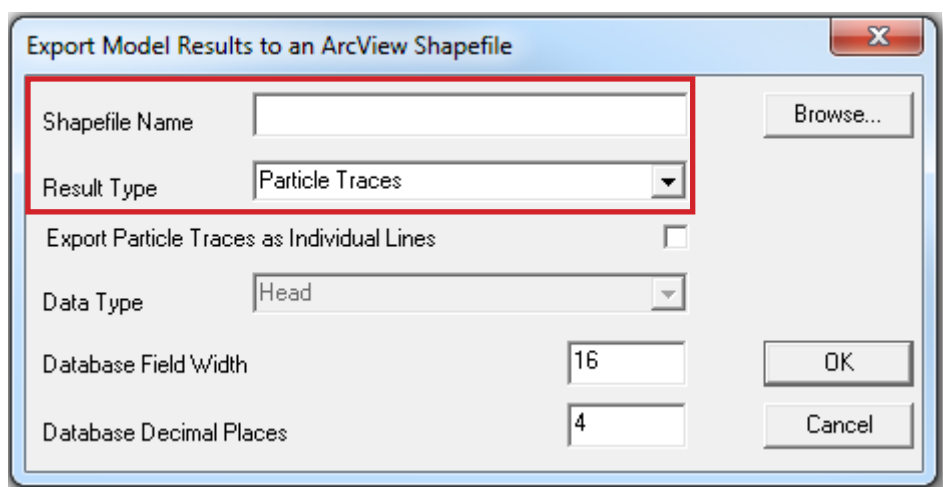**Click on a particle circle to open the Circle Trace Information box**

this example, set the top layer to 1 and the bottom layer to 2. Leave the remaining default settings as shown above.

If the particles are not visible after creating them, make sure that the checkbox Display Particle Starting Locations is selected in Plot > What to Display.

Now that you have created a particle circle, create MODPATH datasets (Model > MODPATH > Create Datasets) and run MODPATH (Model > MODPATH > Run MODPATH). Just as for running MODFLOW in Vistas, you can view the error/warning file by clicking "yes" in the pop-up window. Once MODPATH is finished running, particles can be displayed by selecting the checkbox Display Particle Traces in Plot > What to Display.

To export particles, go to File > Export…, type in a file name, select the appropriate file type (such as a shapefile, .shp), and save. If exporting as a .shp file, the next pop-up window will allow you to name the file and set the type of output to export (Result Type: Particle Traces).

> **TIP:** If working with exported particle traces in ArcMap, be sure to correct the location of the particles back to the real-world coordinates at the center of your well of interest. If not, your particles will be offset by the same distance you moved them in Vistas to center them in the middle of the model cell.

Export particles to shapefile

Calculating drawdown (comparing scenarios)

Drawdown calculations are performed to compare one set of water levels to another, such as comparing predevelopment water levels to current-day conditions, or evaluating the impact of adding or removing wells. Regardless of the objective, it is important to follow the steps outlined here.

First, save and run a version of the model that reflects initial drawdown conditions. For example, if interested in determining current drawdown compared to predevelopment, then predevelopment drawdown (no wells) represents initial conditions. If interested in determining drawdown under an alternative scenario (say, wells removed or wells added), then the existing model represents initial conditions. (Refer to the section on wells in "Viewing and Editing Model Features" for more information about changing these features in the model.)

TIP: Before running the initial-conditions version, be sure to uncheck the box "Set Head to HDRY in cells with small saturated thickness (IPHDRY)" under Model > MODFLOW2005 > Options > LPF Options. Running the initial-conditions version of the model will generate a heads (.hds) file which will serve as starting heads in the subsequent model run under changed conditions.

When the initial-conditions run is complete, open a version of the model that reflects the changed drawdown condition (for example, well(s) added or well(s) removed). Next, go to Model > MODFLOW > Package Options > Initial Heads, set the initial head location to Read Heads Directly from Binary Head—Save File, and browse for the corresponding file name. Select the .hds file that corresponds to the initial drawdown condition created earlier. As with the initial-conditions version, before running the model you'll need to uncheck the box Set Head to HDRY in Cells with Small Saturated Thickness (IPHDRY). The drawdown comparison will only work if this HDRY option is unchecked for both the initial- and changed-condition runs.

Now create datasets, run the model, and import results for the changed-conditions version. Be sure to check the box to import the drawdown file when importing results. Once results are imported, go to Plot > What to Display, check the box Display Contours of, and select Drawdown from the drop-down menu to display contours of drawdown. To display a color flood of drawdown, check the box Display Color Flood of and select Drawdown from the drop-down menu.

Transient simulations

The transient model simulates the year 2014. It uses one stress period for each month of the year, for a total of 12 stress periods representing 365 days. The number of time steps per stress period ranges from 2 to 5, with the relative length corresponding to the number of days in the month. Recharge and pumping rates vary monthly. The model is described in additional detail in the modeling report.

Be aware that transient simulations require significant model setup, and might require run times lasting several hours. When editing a transient simulation, model stability and convergence can be issues depending on the specific problem being investigated.

Best practices for successful transient runs

The following tips should help in setting up successful transient runs:

- Use initial heads as near to the final solution as possible. This should not be an issue unless major changes are made to the model.

- When editing model parameters, be sure to properly set up all transient data for **each stress period**. Failing to do this is a common mistake and will lead to model crashes. Properties with transient data include well pumping, recharge, streamflow routing, and boundary conditions. See "Editing Transient Model Parameters" for details.

- If using the model to plot hydrographs, use the GAGE Package to capture stream flow and other mass balance data rather than saving entire model mass balance at every location. (See "Using GAGE Package for Stream Hydrographs" for more details.)

- When using the GAGE Package, run Vistas using the native executable file and correct file settings as described in "Using GAGE Package for Stream Hydrographs."
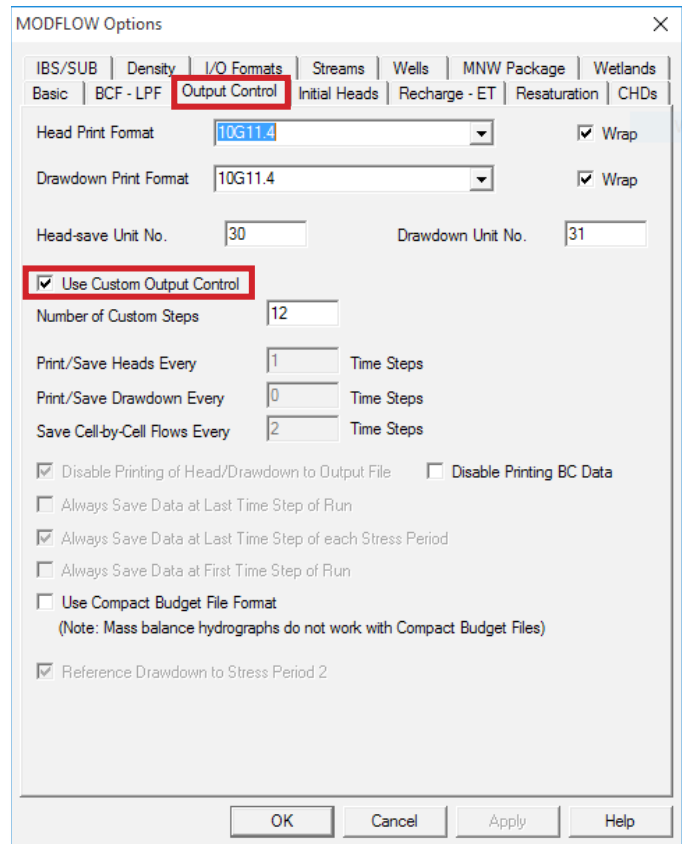
Technical setup

Initial heads

A transient run should begin with initial (starting) heads taken from a previous successfully converged steady-state or transient run. During the initial run the Set Heads to HDRY in Cells with Small Saturated Thickness (IPHDRY) box in Vistas under MODEL > MODFLOW2005 > OPTIONS must be unchecked. Files built with this box checked are not suitable for starting heads, and models using such files for initial heads will probably not converge. Unfortunately, this is the opposite of what is needed for using MODPATH (see "Particle Tracking Using MODPATH"). Setting initial heads is the same for transient and steady-state. Select Model > MODFLOW > Package Options > Initial Heads, set the Initial Head Location to Read Heads Directly from Binary Head—Save File, and browse for the corresponding file name from the initial run.
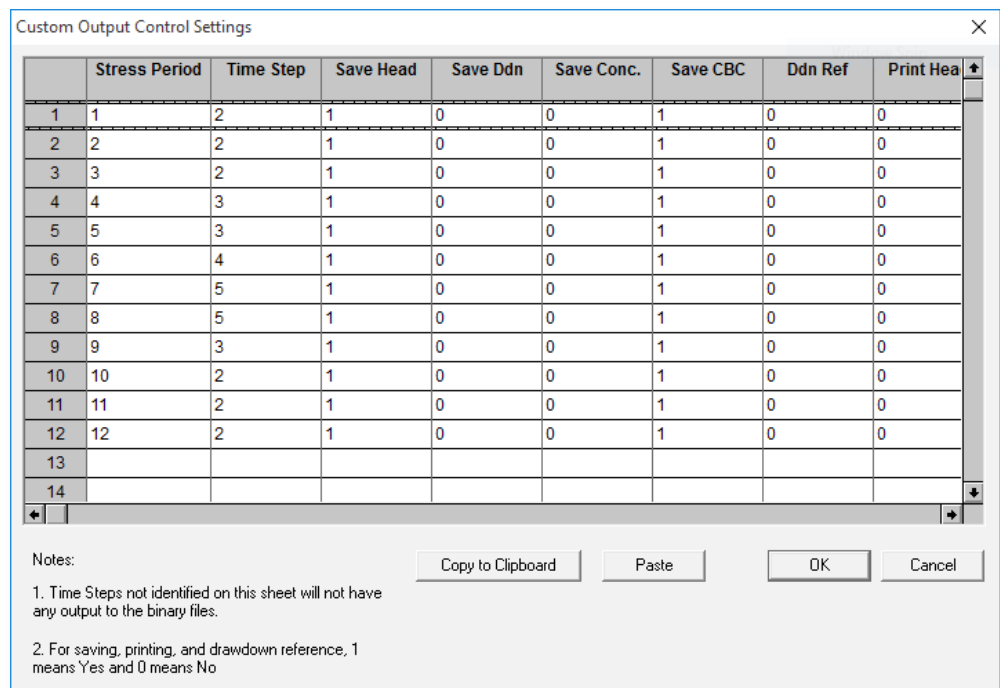
Output control

Output control is used to determine how often data is saved. You should not have to modify this unless you wish to save data more or less frequently. In a transient run, the files generated by Vistas are quite large if the model saves data from every time step. For example, the LPR model has a total of 38 time steps, and if the heads and flows from each time step were saved it would be equivalent to the combined output of 38 steady-state models. To cut down on this output we use custom output control. As seen in the first screenshot, the Use Custom Output Control box has been selected in the Model > MODFLOW > Package Options > Output Control tab. The settings for output control are set to save the heads and mass balance at the end of each stress period. This is shown in Model > MODFLOW > Custom Output Control (see screenshot below).

Check box for Custom Output Control

Sample custom output control settings

	Stress Period	Time Step	Save Head	Save Ddn	Save Conc.	Save CBC	Ddn Ref	Print Hea
1	1	2	1	0	0	1	0	0
2	2	2	1	0	0	1	0	0
3	3	2	1	0	0	1	0	0
4	4	3	1	0	0	1	0	0
5	5	3	1	0	0	1	0	0
6	6	4	1	0	0	1	0	0
7	7	5	1	0	0	1	0	0
8	8	5	1	0	0	1	0	0
9	9	3	1	0	0	1	0	0
10	10	2	1	0	0	1	0	0
11	11	2	1	0	0	1	0	0
12	12	2	1	0	0	1	0	0
13								
14								

Notes:

1. Time Steps not identified on this sheet will not have any output to the binary files.

2. For saving, printing, and drawdown reference, 1 means Yes and 0 means No

Running the transient model

The model can be run as described for the steady-state model in "Running the Model," unless you are using the GAGE Package, which requires the name file be modified (see "Using the GAGE Package for Stream Hydrographs").

This section describes a few additional methods to run the model, which can be used for either transient or steady-state. They are included here as they are often useful for longer transient runs.

By clicking on the USGS executable (.exe) file

The model can be run directly using the USGS MODFLOW executable file. This method allows the user to run the model without Vistas. Save the executable file (MODFLOW-NWT_64.exe) in your working directory, then double-click on it, and enter the name file in the DOS command line. The model will begin running as described in "Running the Model."

By using a Windows batch (.bat) file

Another way to start the model is by using a batch file. This method is handy if you are running a model repeatedly. To use this method, you need to prepare a Windows batch file by opening a text editor and saving with the extension .bat.

Here is an example, called Runtransient.bat:

MODFLOW-NWT_64.exe Usethis_Trans_predevel.nam

MODFLOW-NWT_64.exe Usethis_Trans_2014_11_29_15.nam

In this case, the name files refer to two different models to be run. Assuming the name files have been set up correctly, this batch file will run two models, one after the other. To run the model this way, double-click on the batch file name. This will execute the model and add the name file automatically, saving some typing. This also allows you to run several models in a row while you are away from your computer.

Monitoring a transient run

This section describes how to monitor transient model runs that use the USGS executable file. Steady-state runs may be monitored in the same way, although they often finish running more quickly and monitoring is not needed.

After running the model using the USGS executable (.exe) file, the DOS window will close when the model either finishes successfully or crashes. If it closes quickly (within a minute or so) there is likely an input problem. Successful transient runs might take as long as 1–2 hours depending on the simulation. To abort a run, press control-C in the DOS command line.

Because of these long run times, and because there is little screen feedback from the model, you might wonder whether it is actually running.

There are three ways to track the progress of the run:

- Look at the stress period and time step numbers in the DOS window. The numbers should gradually increase, changing every few minutes.

- Look at the size of the list (.lst) file; it should gradually increase while the model is running.

- Use Windows Explorer (or another file viewer) to open the .lst file, then scroll to the bottom to see what the model is doing. Opening this file will not interrupt the model run, but you will need to refresh (close and reopen) the file to track ongoing progress. Refreshing the file is particularly important if the model is taking a long time between time steps. If the model is not converging during the time steps, you can determine this by checking to see where the model is slowing down, such as near a model cell with a problematic well or stream section. If you spot this behavior, there is usually no point in continuing; it's best to abort the run (control-C) and fix the problem. For help diagnosing problems, see "Additional Resources."

Sometimes the model will crash, and might list some errors. Again, looking in the .lst file is a good way to track down errors. Often crashes are due to errors or omissions in the name file.
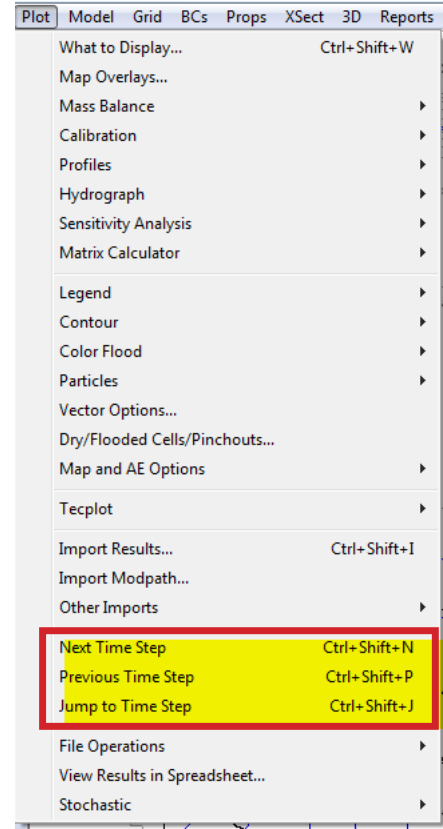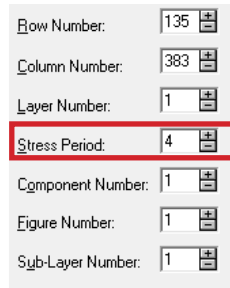
Looking at transient output

Transient output files (heads, drawdowns, mass balance) can be imported into Groundwater Vistas similar to steady-state runs, with the difference that you must specify the stress period and time step desired for viewing. Be aware that import might be slow because the output files can be very large, and Vistas has to sort through the large files to load the correct time step. This model has 12 stress periods with up to 5 time steps each. You can choose the time step to display using the browse function under Import Model Results and highlighting the time desired.

Vistas also has a handy feature under the Plot menu that lets you switch between time steps (see screenshot at right).

Note that changing the stress period on the left sidebar only changes the displayed input values, not results.

Changing the stress period in the sidebar changes displays input values only, not results
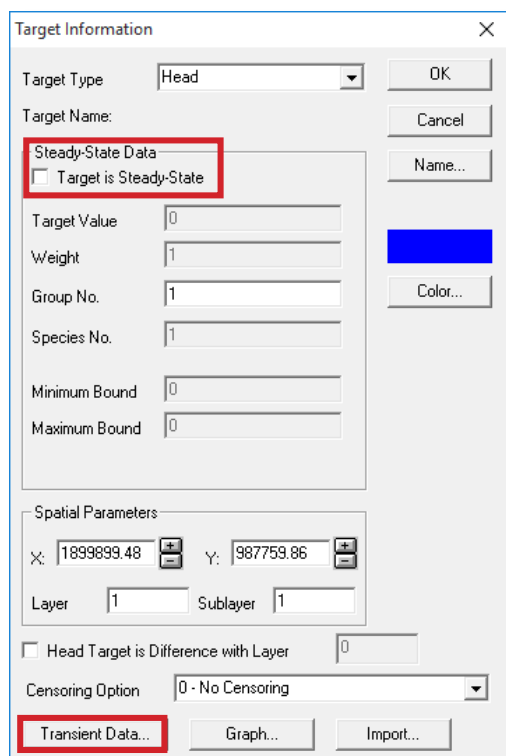
A handy shortcut for switching between time steps

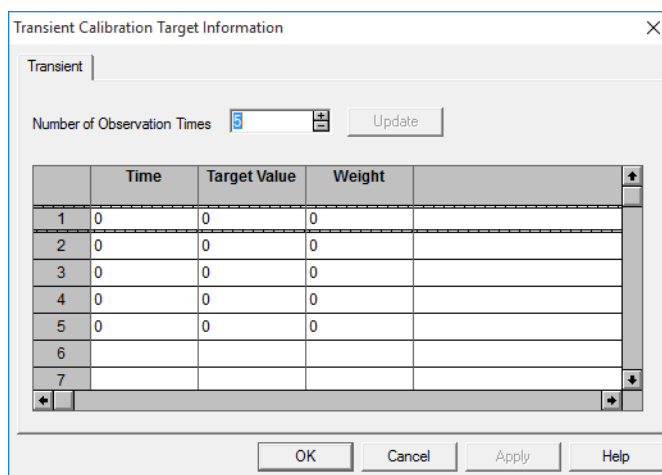**When importing results, choose the time step using the browse function**

Plotting hydrographs

Using Vistas targets or monitoring wells for head

For transient runs, users will probably want to generate hydrograph plots of drawdown, water levels, or stream flow at specific locations. For monitoring head, this may be done by adding a target or monitoring well under the Analytic Element window prior to running the model (AE > Target or AE > Well). A target is typically a location where head or flux has been measured, and can be compared graphically to model results. Enter transient data by unchecking Target is Steady-State and clicking on the Transient Data button (see screenshot). You may also import data using the Import button.

Enter transient data by unchecking "Target is Steady State" box, then clicking the Transient Data button
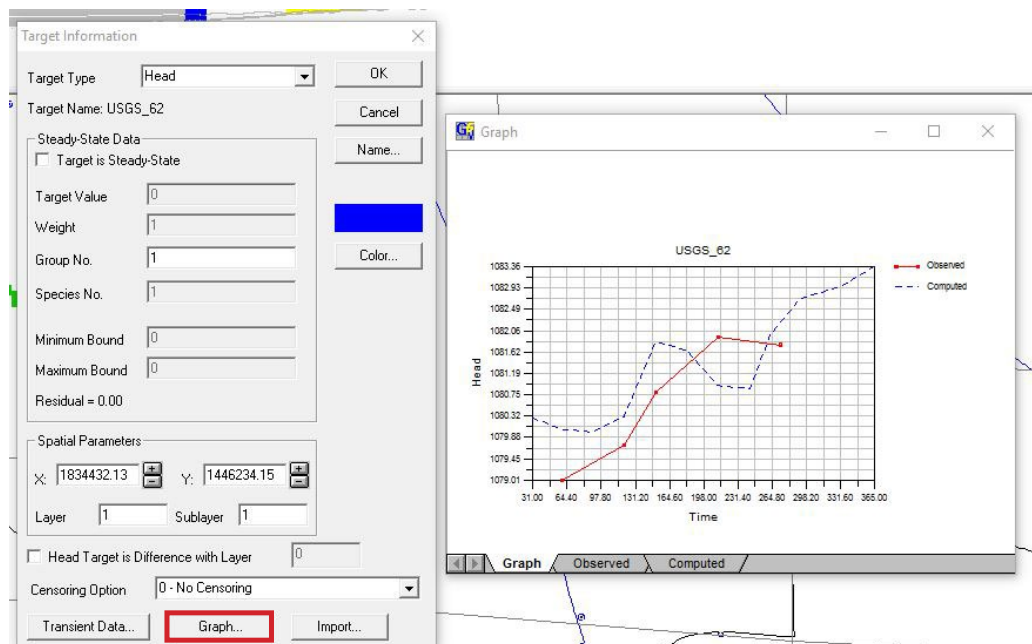

Transient data entry area

After loading the model results you can click on a target to see and/or export a hydrograph. To do this, first click on the target; this brings up the target menu. Then click on Graph to bring up the hydrograph (see screenshot below). Right-click on the graph to see export options.

Alternatively, you can add a well to monitor head (AE > Well). Check Monitor Head/Concentration vs. Time (see screenshot). Once you have run the model, view results using Plot > Hydrograph > Monitoring Well.

Click on a target to view its hydrograph; right-click on the graph to export

Monitor head by checking highlighted box

Using GAGE Package for stream hydrographs

We recommend using the GAGE Package when plotting transient hydrographs of flow or stage. For hydrographs, all the mass-balance data must be saved for every time step. Saving such data for all locations and time steps will generate enormous cell-by-cell (.cbb) files. To avoid this problem, the GAGE Package will save the head or flow data only at specified stream nodes. Vistas includes an option to prepare the GAGE file, but writes all the data to a single large file. We have set up a gage file (Usethis_Trans_2014_baserun.gag) that saves data at four specific points along the Little Plover River and stores each data set in a separate file. The four points represent gages with discharge data at Kennedy Avenue, Eisenhower Avenue, Highway 51, and Hoover Avenue.

A gage file may have the extension .gag or .dat. The screenshot at right shows the content and formatting of a typical gage file.

To use the GAGE Package, the gage file and the specified output files must be indicated in the name (.nam) file. We have provided an example name file (Use_for_Gage_package_Trans_2014_7_24_16.nam) for this purpose. You may use these files for running the Gage Package unless you change the root file name. The root file name can be
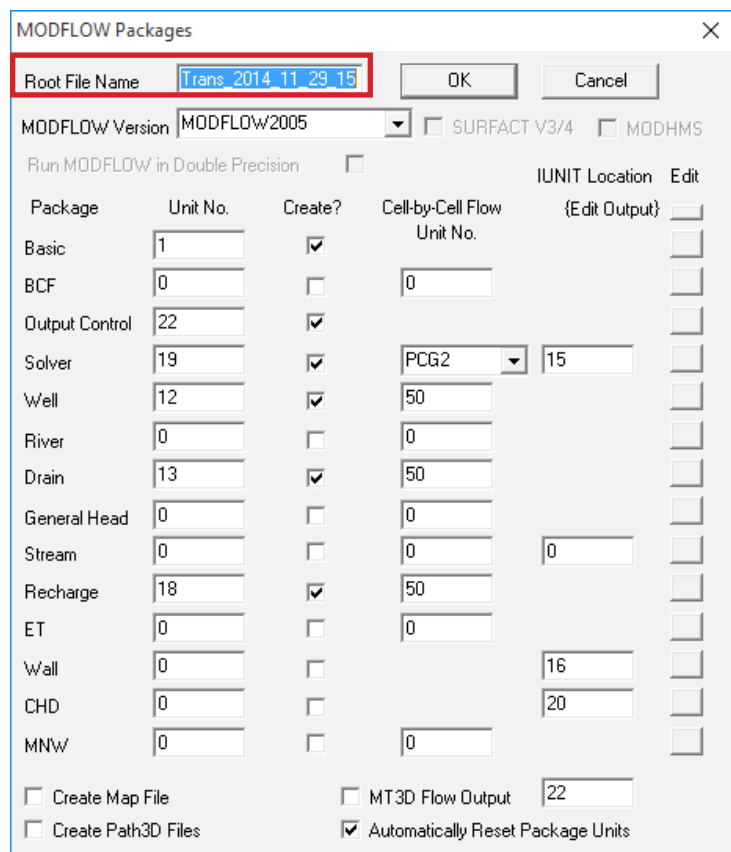
SAMPLE GAGE FILE
(see MODFLOW documentation for details)

found in Model > MODFLOW > Packages (see screenshot). If you change the root file name, you will need to create your own edited name file (see "Changing the Name File"). The gage file should only be changed if you wish to evaluate a different stream location. If you choose to modify the gage file, we recommend reading the MODFLOW documentation here: http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/gage.htm.
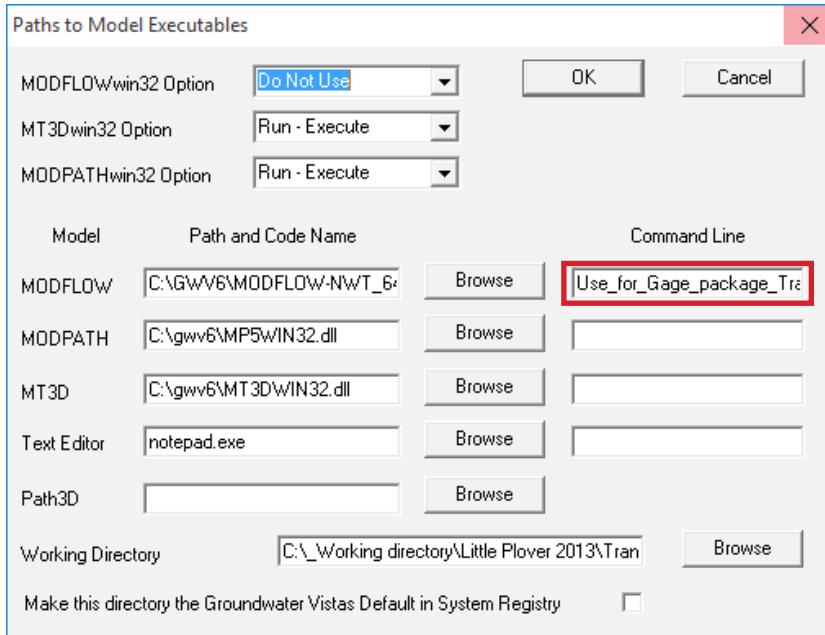
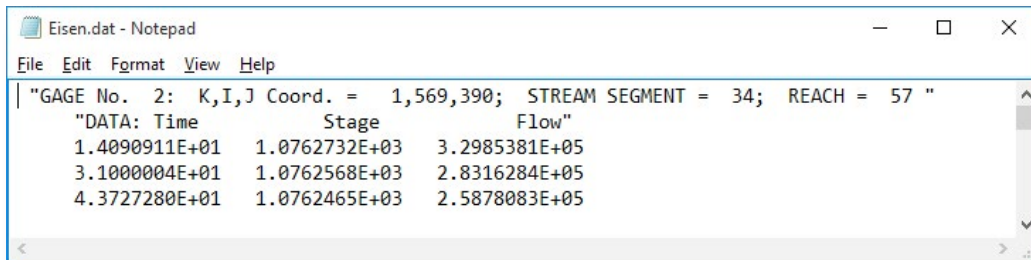

View or edit the root file name

The output files for the four gage locations will be saved in the working directory. The screenshot below shows the partial contents of the Eisenhower gage output file (Eisen. dat). You can use another program, such as Grapher or Excel, to plot the hydrographs.

NOTE: When using the provided gage file, you must run the model using the USGS executable (.exe) file as you would for steady-state. However, **you must reference the modified name file in the command line** under Models > Paths to Models (see screenshot).

Reference modified name file when using GAGE Package

Partial contents of a gage output file

Changing the name file

To change the name file, first create datasets in Vistas (Model > MODFLOW > Create Datasets). This will generate a name file similar to the one shown on the left in the screenshot below. Use a text editor such as Notepad to add the gage file name and appropriate output files. A text file with instructions has been included with the transient model files.

The new name file should look similar to the screenshot (below right), except "Trans_2014_7_24_16" will be replaced by your root file name. The yellow highlight shows the added lines.

Save the edited name file with a different name that you will remember. Every time you create datasets, the original name file will be rewritten. By using a different name, you are able to run various scenarios without changing the name file. However, remember to edit the name file whenever you change the root file name.

The format of the name file is important; make sure the text editor does not change the format. Also, make sure the extension of the edited file is .nam. Some text editors will automatically change it to .txt or .dat; these will not work.

Name file created by Vistas

ABOUT THE ENTRIES

GAGE 28: References gage file, where "28" is the unit number assigned to the file by Vistas. "GAGE" must be written out.

DATA 45-48: Specifies the name of the output file for each gage and unit number (45-48) referenced in the gage file.

Editing transient model parameters

Transient model properties must be set up for all stress periods. Properties with transient data include recharge, streamflow routing, boundary conditions, and well pumping. Basic changes to properties is discussed in "Viewing and Editing Model Features." This section describes how to apply these changes to multiple stress periods. Note that boundary conditions cannot be changed within a stress period.

Changing recharge and boundary conditions

Users are cautioned that any changes to recharge in the transient model must be applied to all stress periods affected. Change the Stress Period box on the left side of the Vistas screen to view and edit the recharge for different stress periods.

Likewise, boundary conditions need to be specified for each stress period affected. Just changing a boundary condition in the first stress period will not work. After any change you need to copy the change to all stress periods using the Vistas procedure: BCs > Modify > Copy Stress Period and then put in the range of stress periods you wish to copy to. The same technique applies to changing properties of streams.
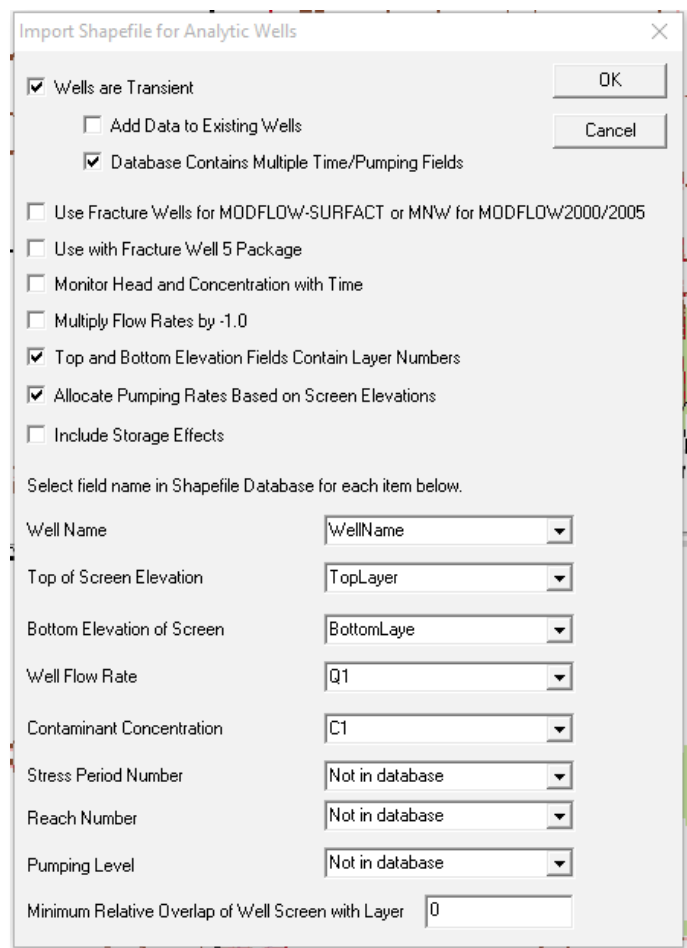

Edited name file

Adding transient wells

For transient runs of the Little Plover model it is best to use the analytic element option in Vistas and import the wells directly from a GIS shapefile (AE > Import > Shapefile). The benefit of entering wells this way is that each well can penetrate one or more model layers as appropriate for its well construction and depth and Vistas will apportion flow to each layer based on the layer transmissivity contrast. Using this method, Vistas writes out the MODFLOW input (.wel file) using standard MODFLOW format, and will thus convert an analytic element in a well that spans multiple model layers into two or more MODFLOW wells, one for each layer. This can be confusing because it will cause the number of wells listed in the MODFLOW .wel file to be greater than the actual number of wells simulated. For example, the transient LPR base model simulated 448 wells, but because some of these wells span multiple model layers, the MODFLOW .wel file lists 621 wells. Vistas will report the actual number of wells by clicking MODEL > Model Information.

The Little Plover model contains several hundred wells, each with a unique pumping record. It is convenient to enter these wells using a shapefile containing the pumping record for each well on a single line. Below is an example of an appropriate format in the .dbf shapefile (such a file can be set up directly from the Wisconsin DNR well database). The following example only shows the first five stress periods, but continues in a similar pattern through the last stress period. There are 12 pumping rates (Q1, Q2, Q3…etc.), corresponding to the 12 stress periods in this run. The C1, C2, C3… columns are for concentration data, not used in the current model, so these columns are set to zero.

To import these data into Vistas, use AE > Import > Shapefile and specify Well. Then browse for the correct shapefile. This will bring up the screen below. Make sure that you understand the data entry process, which can be confusing. In this example, for a model with 12 stress periods, Vistas will look for the pumping data for stress period 1 in column Q1, the concentration in column C1, and will then look for the remaining pumping and concentration data for that well in the following 22 columns in the shapefile database (11 for Q and 11 for C). Be sure to check Vistas after you are done to be sure that the correct number of wells was entered and that the data are what you expect. Just click on any well symbol in Vistas to bring up the data for that well.

Well import menu

FID	Shape *	WellName	TopLayer	BottomLaye	Q1	C1	Q2	C2	Q3	C3	Q4	C4	Q5	C5
146	Point	1013	2	3	0	0	0	0	0	0	0	0	0	0
388	Point	1162	2	3	0	0	0	0	0	0	-33687.4977	0	-55421.3671	0
287	Point	1192	2	3	0	0	0	0	0	0	0	0	0	0
397	Point	1279	2	2	0	0	0	0	0	0	0	0	0	0
343	Point	1302	2	3	0	0	0	0	0	0	0	0	0	0
319	Point	1467	2	2	0	0	0	0	0	0	0	0	-2112.0664	0

Partial contents of example shapefile for transient pumping import

Changing well pumping

To edit transient pumping rates, click on the well and make sure the Pumping Rate is Steady-state box is unchecked. Click on the Transient Data button to enter transient pumping rates (see screenshot).

In the example shown below, the well is pumped at a different rate for each of the 12 stress periods. This particular well is not pumped at all during stress periods 1 to 4 (representing January through April). Beginning in stress period 5 (May) the well is pumped during the summer months, but is idle again in the fall (not shown in screenshot).

To edit transient pumping rates, uncheck the "Pumping Rate is Steady-state" box, then click the Transient Data button
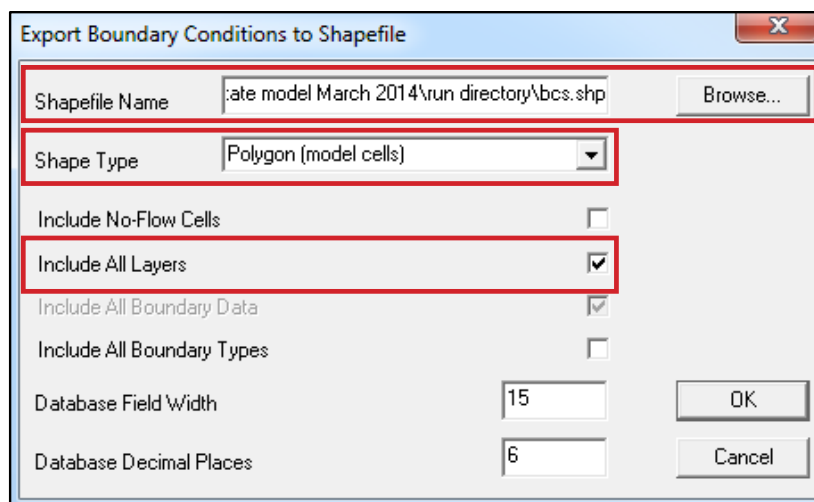

Example data for a well pumped at different rates for each stress period

ArcMap SFR2 baseflow comparison macro

We've created a macro that runs in ArcGIS to calculate streamflow changes at various sites throughout the model. This section describes the macro and how to use it.

What the macro does

This ArcMap macro helps evaluate changes in streamflow between different model runs for any set of locations along the Streamflow Routing (SFR2) network. This is particularly useful when determining cumulative streamflow (baseflow contribution) at multiple stream locations.

Installing the macro

From any current session of ArcMap, open the ArcToolbox, right-click on the ArcToolbox icon in the upper left hand corner of the pop-up window, and select Add Toolbox. Next, select the Flux Calculations.tbx file from the ArcGIS folder included with the model files and press Open. This will add the macro to the ArcToolbox for the open ArcMap session; you'll need to reinstall the macro each time you start a new ArcMap session.

Files needed to run the macro

To run the macro, you'll need to export two shapefiles from Vistas—the initial SFR2 flux conditions (for example, 2010 steady-state) and the changed SFR2 flux conditions (such as wells added). You'll also need a shapefile containing the point locations where you are interested in extracting stream fluxes along the SFR2 network.

Exporting SFR2 data from Groundwater Vistas
Under the tab BCs, highlight "Stream," then click Export > Shapefile (see screenshot).

Export stream shapefiles

Then, from the Export Boundary Conditions to Shapefile window, browse to where you want to save the file and enter a file name. Set Shape Type to "Polygon (model cells)" and check the box "Include All Layers". (For this macro, leave the other boxes unchecked and leave the default values for database fields.) Finally, press "OK" to export the SFR2 data. A pop-up box with the message "Shapefile exported successfully" will appear.
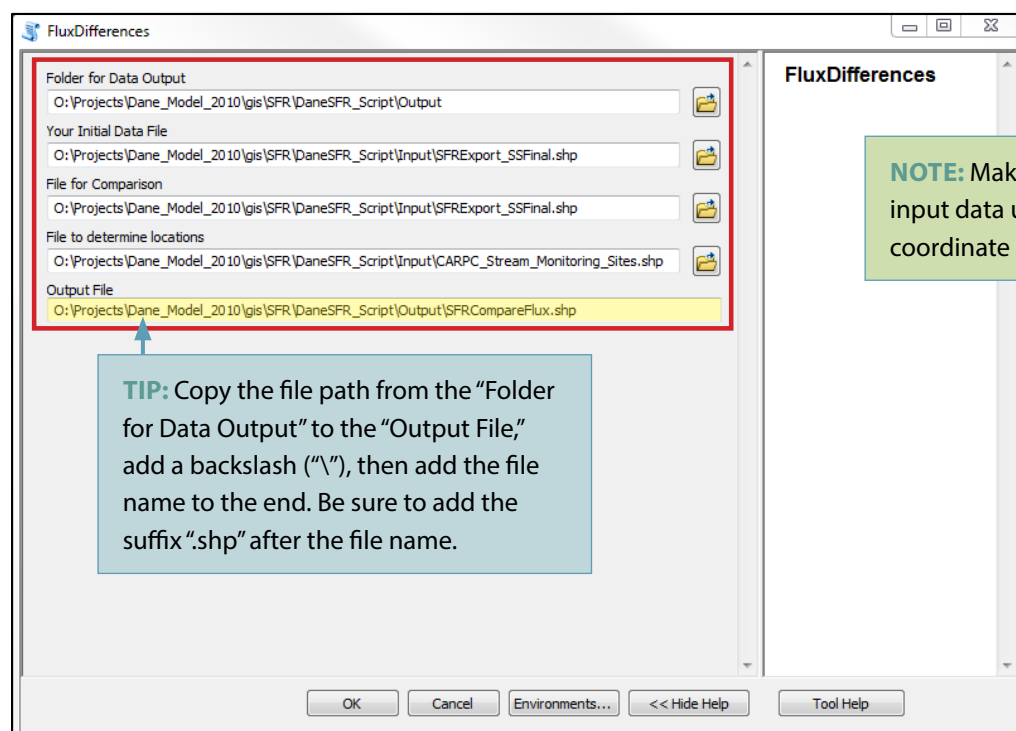

Export boundary conditions

Running the macro

From the ArcToolbox window, open Flux Calculations and double click FluxDifferences to launch the macro (see screenshot).

Select the appropriate output folder and input files as follows:

Folder for data output: Browse for and select the folder you would like the final shapefile to be saved to. Do not select a file in the destination folder, just the folder.

Your initial data file: Browse for the shapefile containing the initial SFR2 flux conditions (for example, 2010 steady-state conditions). Be sure to select the .shp file extension for this shapefile. This is the file to which changed SFR2 conditions will be compared.

File for comparison: Browse for the shapefile containing the SFR2 flux conditions for comparison. This would be the file that represents streamflow under a changed condition such as adding wells.

NOTE: Make sure that all input data use the same coordinate system.

TIP: Copy the file path from the "Folder for Data Output" to the "Output File," add a backslash ("\"), then add the file name to the end. Be sure to add the suffix ".shp" after the file name.

Select files and output folder location, then click OK to run the macro

File to determine locations: Browse for the shapefile containing point locations of the spots you're interested in performing the SFR2 comparison. Make sure the locations of comparison points are located completely within SFR2 model cells.

Output file: Enter the full file path and file name of the output file. This is most easily done by copying the file path of the Folder for Data Output and adding the file name onto the end. Be sure to add a backslash ("\") before the file name and the suffix ".shp" after the file name (see screenshot).

Viewing macro results

To view results, add the newly created shapefile to an ArcMap session and open the attribute table. GWDiff is the difference in groundwater baseflow to each SFR2 cell selected for comparison (equivalent to flux ("F") as described in the "Understanding Streamflow Reporting" section), while SDiff is the difference in cumulative surface water flow of every SFR2 cell upgradient of and including the SFR2 cell selected for comparison (equivalent to "flow," as described in "Understanding Streamflow Reporting"). Units of flow will be in cubic feet per day, the same as the SFR2 data originally exported from Groundwater Vistas.

References

Bradbury, K.R., Fienen, M.N., Kniffin, M.L., Krause, J.J., Westenbroek, S.M., Leaf, A.T., and Barlow, P.M., 2017, A groundwater flow model for the Little Plover River basin in Wisconsin's Central Sands: Wisconsin Geological and Natural History Survey Bulletin 111, 82 p.

Prickett, T.A., and Lonnquist, C.G., 1971, Selected digital computer techniques for groundwater resource evaluation: Illinois State Water Survey Bulletin 55, 62 p.

UW Extension

University of Wisconsin-Extension

Our Mission

The Survey conducts earth-science surveys, field studies, and research. We provide objective scientific information about the geology, mineral resources, water resources, soil, and biology of Wisconsin. We collect, interpret, disseminate, and archive natural resource information. We communicate the results of our activities through publications, technical talks, and responses to inquiries from the public. These activities support informed decision making by government, industry, business, and individual citizens of Wisconsin.