

The 2016 Groundwater Flow Model
for Dane County, Wisconsin

User's Manual



Bulletin 110-supplement • 2016

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

Contents

Introduction	1	Transient simulations	19
Additional resources	1	Best practices for successful transient runs	19
Background information and settings	2	Technical setup	19
Model layers	2	Initial heads	19
Model units, orientation, and GIS options	3	Files needed for running the transient model	19
Steady-state and transient models	3	Changing the name file	20
Technical setup	4	Output control	22
Newton Solver	4	Running the transient model	22
Directories	4	By running the USGS executable file in Vistas	22
Map files	4	By clicking on the executable (.exe) file	23
Running the model	5	By using a Windows batch file (.bat)	23
By running the USGS executable file in Vistas	5	Monitoring a transient run	23
By running the .dll file in Vistas	6	Looking at transient output	24
Viewing model results	7	Plotting hydrographs	25
Run time	7	Using Vistas targets or monitoring wells for head	25
Model output	7	Using GAGE package for streams or lake stage	27
Viewing and editing model features	8	Editing transient model parameters	28
Lakes (LAK3 package)	8	Changing recharge and boundary conditions	28
Streamflow routing (SFR2 package)	8	Changing well pumping	28
Wells (MNW2 package)	9	ArcMap SFR2 baseflow comparison macro	29
Adding a new well	9	What the macro does	29
Editing an existing well	10	Installing the macro	29
Removing an existing well	10	Files needed to run the macro	29
Groundwater contribution to wells by layer (borehole flow)	11	Exporting SFR2 data from Groundwater Vistas	29
Springs	11	Running the macro	30
Recharge	11	Viewing macro results	31
Understanding streamflow reporting (SFR2 package)	12	References	31
Particle tracking using MODPATH	14		
Calculating drawdown (comparing scenarios)	18		

Suggested citation:

Bradbury, K.R., Parsen, M.J., and Fehling, A.C., 2016,
The 2016 Groundwater Flow Model for Dane County,
Wisconsin—User’s Manual: Wisconsin Geological and
Natural History Survey Bulletin 110–Supplement, 31 p.

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.



Published by and available from:

Wisconsin Geological and Natural History Survey

3817 Mineral Point Road ■ Madison, Wisconsin 53705-5100
608.263.7389 ■ www.WisconsinGeologicalSurvey.org
Kenneth R. Bradbury, Director and State Geologist

ISSN: 0375-8265

ISBN: 978-0-88169-991-3



Introduction

This document provides basic instructions and recommendations for running the steady-state and transient versions of the 2016 Dane County groundwater flow model using the Groundwater Vistas (Vistas) user interface. The steady-state model was calibrated to reflect average groundwater conditions between 2006 and 2010. There are also two transient models—one is an example of a new well pumping under average conditions, and the second is calibrated to the drought period during June and July 2012. This model has been significantly improved since release of the 1996 model (Krohelski and others, 2000).

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, *The 2016 Groundwater Flow Model for Dane County, Wisconsin* (Parsen and others, 2016).

The 2016 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 Dane County model: <https://uwmadison.box.com/dane-groundwater-model>

This model was last modified using version 6.74 build 37 of Vistas, so users should update to at least this version of Vistas before attempting to run the model.

NOTE: This guide assumes the user has a basic knowledge of MODFLOW and Groundwater Vistas, and is not intended to replace user manuals for these programs. Refer to the list of additional resources for links to recommended guides.

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 new model uses a 12-layer system. Figure 1 identifies each layer and shows how they relate to the previous model’s layers as well as to the general bedrock geology of

Dane County. The thicknesses and vertical extents of the new model layers are not necessarily coincident with the stratigraphic unit formations.

Note that all model layers are treated as continuous across the model domain. In places where specific layers are absent, they are referred to as “pinchout” layers. The thicknesses of pinchout layers are set to 0.2 ft and made effectively transparent by assigning hydraulic conductivity values (K_v , K_x , and K_y) as equal to those of the first underlying layer that does not pinch out.

GENERAL BEDROCK STRATIGRAPHY				GROUNDWATER FLOW MODEL				
Age		Stratigraphic name		Model layers, names				Type
Era	Period	Group	Formation	1996 model		2016 model		
				1	Sand and gravel	1	Unlithified I (fine-grained lacustrine deposits within the glacial Lake Yahara area)	aquifers
						2	Unlithified II (till and meltwater stream deposits)	
Paleozoic	Ordovician	Sinnipee	Maquoketa	2	Upper bedrock	3	Upper bedrock	
			Galena					
			Decorah					
			Platteville					
		Ansell	Glenwood					
			St. Peter					
	Prairie du Chien							
	Cambrian	Trempealeau	Jordan			4	Jordan	
			St. Lawrence			5	St. Lawrence	
		Tunnel City	Lone Rock, Mazomanie				6	
							7	Tunnel City—Mid (fracture layer)
							8	Tunnel City—Lower
Elk Mound		Wonewoc		9	Wonewoc—Upper			
			10	Wonewoc—Lower (fracture layer)				
		Eau Claire	3	Eau Claire	11	Eau Claire	aquitard	
		Mount Simon	4	Mount Simon	12	Mount Simon	aquifer	
Precambrian		Various unnamed units		No-flow boundary				

Figure 1. Bedrock stratigraphy and corresponding layers in the 1996 and 2016 groundwater models. (General bedrock stratigraphy was adapted from *Preliminary Bedrock Geology of Dane County, Wisconsin*: Wisconsin Geological and Natural History Survey Open-File Report 2013-01.)



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.

There is no grid rotation.

Within Dane County, which represents the near-field of the model domain, each cell size is 360 x 360 ft. Beyond Dane County, in the far-field of the model domain, grid cell sizes increase using a multiplier of 1.4. North and south of Dane County, this multiplier is applied to the y-dimension (row height) resulting in taller model cells, while east and west of Dane County, the multiplier is applied to the

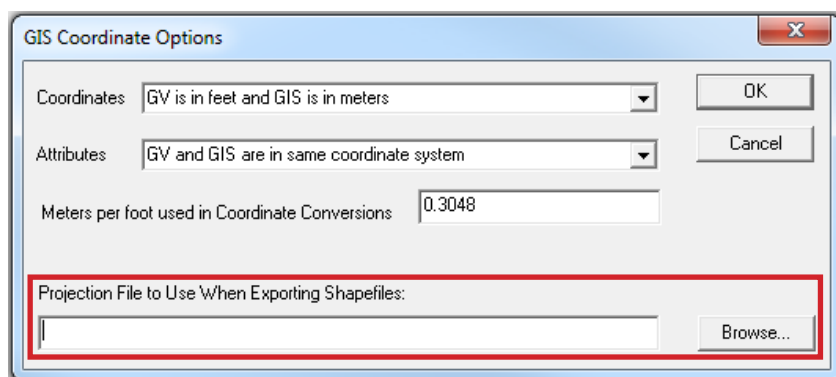
x-dimension (column width) resulting in wider cells. Cells to the northwest, northeast, southeast, and southwest increase in size in both the x and y directions.

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 Dane County groundwater model was calibrated both to steady-state and transient conditions. The steady-state model represents average equilibrium conditions where heads or fluxes do not change over time, while the transient model is capable of accounting for changes in head or fluxes over time. The steady-state model was developed first and is generally calibrated to average hydrogeologic conditions (such as pumping rates, recharge rates) as they existed between 2006 and 2010. The steady-state model was calibrated by varying recharge, hydraulic conductivity, and stream and lakebed properties. It also served as the starting point for transient calibration, which simulated the drought conditions during June and July 2012. During transient calibration, recharge was assumed to be zero and only storage parameters (specific storage and specific yield) and lake surface evaporation were varied to match observed data.

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



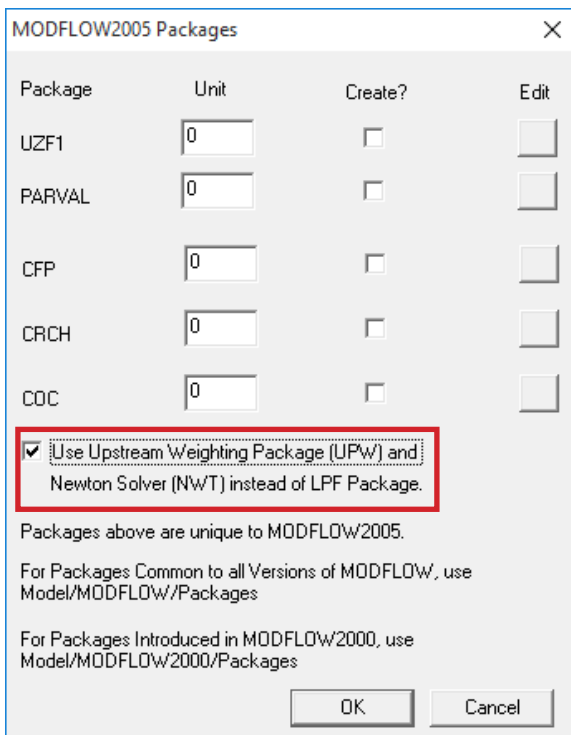
Orient the model with map files when exporting to GIS.



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, and is essential for the complex 12-layer Dane County model. In Vistas, this solver is included in Model > MODFLOW2005 > Packages. While MODFLOW-NWT is already selected in the 2016 Dane County 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.



Check box to activate Newton Solver

The solver parameters are set under the Model > MODFLOW2005 > OPTIONS > NWT General tab. However, these options have been optimized for best model performance and 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 (PCG2, etc) 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 > Path 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.

Map files

The 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.

The three preloaded maps files:

- Dnbndw.map county boundary
- Dnhydrow.map surface water features
- Dane_roads_2012.map roads and highways, current as of 2012

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 unselecting the box Display Base Maps. For a map to be viewable it must be set to Draw in the Plot > Map Overlays window.

TIP: The roads map is quite detailed and can be slow to load if you are looking at the whole county. We generally turn it off when zoomed out.



Running the model

There are two common ways of running the Dane County 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.

NOTE: In parts of the county, particularly in the unglaciated Driftless Area, large regions of model layers 1–3 are above the water table and thus unsaturated. This will produce the following warning: “...these cells will start out dry...” when the model files are written; the warning can be ignored.

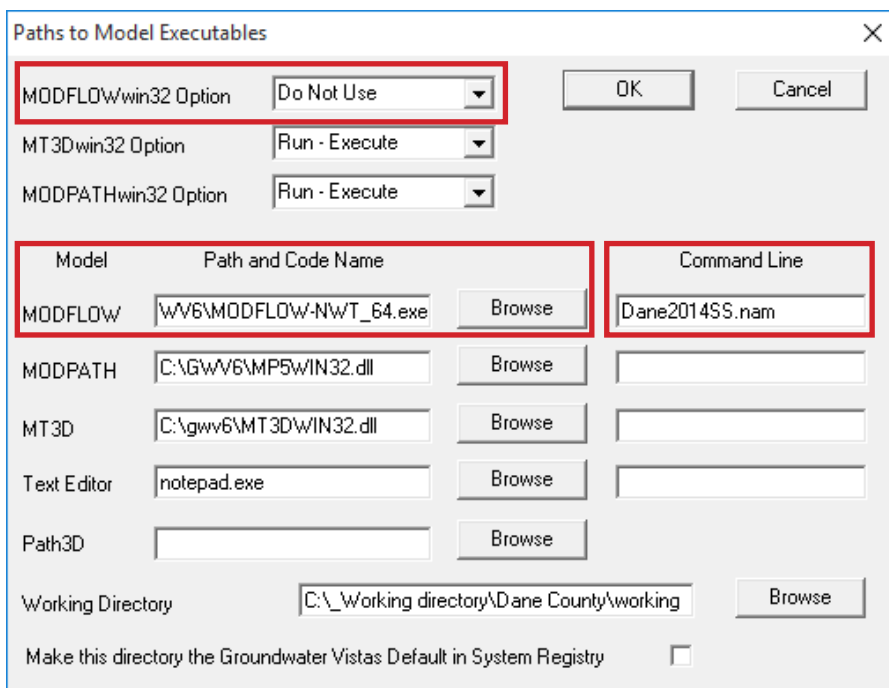
TIP: We recommend using the latest version of the USGS NWT solver (MODFLOW-NWT). As of January 2016, this was version 1.0.9. The file is available from the USGS: water.usgs.gov/ogw/modflow-nwt

By running the USGS executable file in Vistas

The first way to run the model is in Vistas using the executable file from USGS (MODFLOW-NWT.exe or MODFLOW-NWT_64.exe). This method requires that you change two simple 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 .exe file from USGS 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.



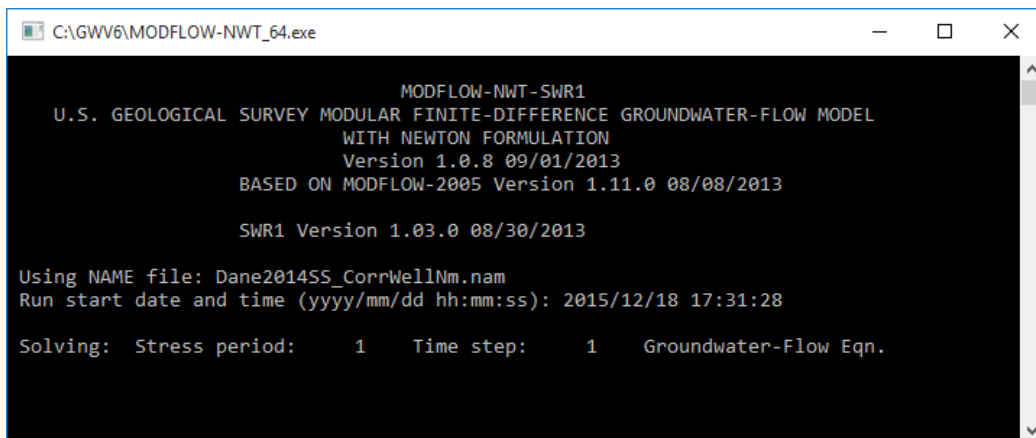
Settings used to run the USGS executable file



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 pop up in Vistas immediately: “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.

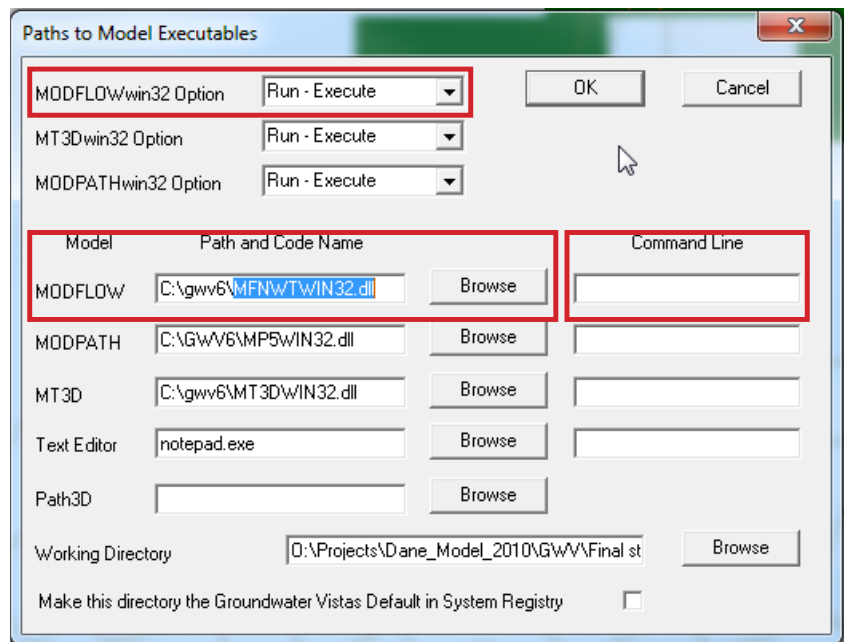


DOS window showing the model running

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 lock up your computer making multitasking very slow. In addition, it does not use the official USGS solver so may give slightly different results than 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 the “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.



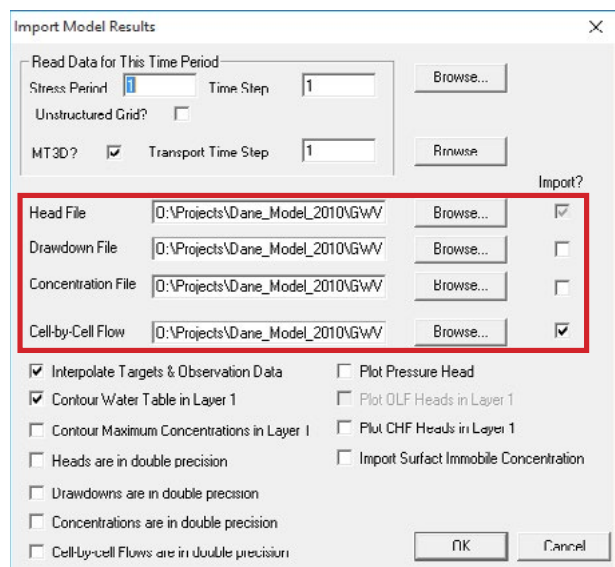
Settings used to run the .dll file in Vistas

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



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 fluxes, you must import the cell-by-cell flow file.



Select results to import for viewing

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 Dane County model generally runs in under 5 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, in addition to the binary files (.hds, .ddn, .cbb) that can be imported into Vistas.

Output files:

File type	Extension	Details
List	.lst	Standard MODFLOW output file with the model budget (including lake budgets) at the bottom.
BYND	.byn	Reports node-by-node flows in or out of each multi-node well, along with the calculated head inside the well, and a seepage elevation if any of the nodes go dry. Important for checking to see that the wells are operating as expected and for evaluating cross-formation flow through well bores.
QSUM	.qsm	Also reports well-by-well flows but totals the flow for each well and gives the average head for the well.

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....



Viewing and editing model features

Lakes (LAK3 package)

The model simulates the major Dane County lakes (Yahara chain plus Fish and Crystal Lakes) using the MODFLOW LAK3 package. This allows the stage, volume, and discharge (for lakes with outlets) to fluctuate during model runs, and also allows the lakes to exchange water with the groundwater system and with routed streams (described below in the “Streamflow Routing” section). In the model input and output, the lakes are numbered as follows:

Model lake num.	Lake
1	Mendota
2	Wingra
3	Monona
4	Waubesa
5	Kegonsa
6	Fish
7	Crystal

A summary of simulated lake parameters, including a water budget for each lake and lake stage, is printed at the bottom of the .lst output file just above the overall model budget. If the Boundary Condition (BC) button has been activated, and cell-by-cell flow has been imported into Vistas, you can hover the cursor over each lake cell to see the simulated stage and flux (F). The data for the node appears in the lower left corner of the screen. Negative flux values indicate flow into the lake (from groundwater); positive flux values indicate flow out of the lake (to groundwater) in units of cubic feet per day. To see the actual flow rate at the inlet or outlet of a lake, hover the cursor over the SFR2 cell that is directly upstream (lake inlet) or downstream (lake outlet) from the lake.

NOTE: Altering the lake parameters may cause model instability. The lake package can respond only to parameters input into the model, such as precipitation, lakebed properties, and groundwater pumping. The Dane County lakes and the Yahara River system also respond to human-made changes (lock and dam settings) and hydraulic changes (weed growth, weed cutting, dredging, backwater effects) that are not directly simulated by the model. The model can be used to investigate these effects, but changing lake and stream parameters should be done with great care and with an understanding of what each parameter does.

Streamflow routing (SFR2 package)

All major streams in Dane County are routed using the streamflow routing (SFR2) package in MODFLOW. The streams are dynamically connected to the lake system to allow groundwater exchange between the two. 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 and 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.

Where continuous human-made discharges to streams are known to occur (such as the Nine Springs sewage treatment plant discharge to Badfish Creek and Badger Mill Creek), average discharge flows are added to the streams at appropriate segments. These types of additional SFR2 flows can be manually modified in the Model > MODFLOW2000 > SFR2 Segment Data... window in the “flow” column. This may be important if you are interested in making pre-development model runs (remove all sewerage discharge flows) or performing other scenarios involving man-made changes to streamflow. This process can be rather tedious because you may need to scroll down through the segment data window to add, modify, or delete individual flow data. For a list of the segments corresponding to all wastewater treatment plant discharges, see appendix 2 in the Dane County model report. For more information on SFR2, see the section “Understanding Streamflow Reporting.”



Wells (MNW2 package)

The model uses the MODFLOW 2000 Multi-Node Well (MNW2) package. This option allows borehole flow to occur between model layers connected by open boreholes even if the well is not being pumped.

To view a well, the Analytic Element (AE) button must be selected and you need to be in a model layer corresponding to the open, or uncased, interval of the desired well. Then, click on the well icon; when the Well Information window opens, select the Basic Data tab. You can see the Multi-Node Well option is checked next to “Use as Fracture Well (FWL4 or 5) or Multi-Node Well (MNW) or CLN (MF-USG)”.

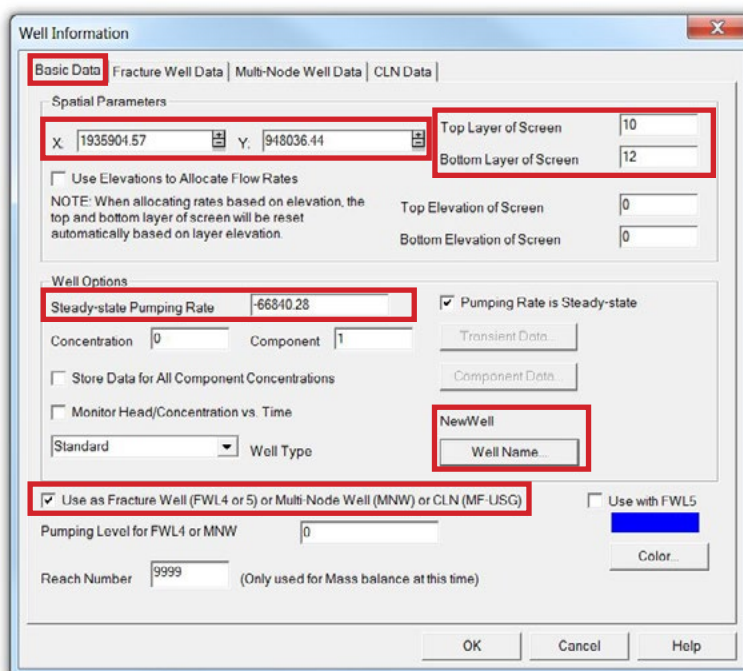
When this option is active then additional parameters are entered on the Multi-Node Well Data tab. These ordinarily do not need to be changed but must be entered if adding a well to the model.

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, click AE > Well and activate the tool. 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. Be sure to click on each of the tabs and complete the information outlined below.

Basic Data tab: 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 groundwater system, while a positive pumping rate represents an injection. Pumping rates must be entered in cubic feet per day so a unit conversion is typically necessary. In this tab you can also name the well. Check the box “Use as a Fracture Well (FWL4 or 5) or Multi-Node Well (MNW) or CLN (MF-USG)” to activate MNW2 functionality. Keep the default settings for all other fields.

NOTE: If pumping causes an individual node to go dry when using the MNW2 package, the model will “turn off” pumping from that node. In other words, MODFLOW adjusts the well discharge rate to sustain pumping. This means that the final simulated pumping rate from an individual well might be less than what was specified in the well input file. There is no obvious warning when this occurs, but it can be easily seen in the .byn file (described in “Model Output”). Check this file to verify that the simulated pumping equals the desired pumping.



Adding a new well—Basic Data tab settings



Fracture Well Data tab: All fields should be left as default.

Multi-Node Well Data tab: Set the “Friction Loss Coefficient (Skin or Rskin)” to “2”, “Well Loss Options (LOSSTYPE)” to “Skin Corrections”, and “Hydraulic Conductivity of Skin (Kskin)” to “10”, as shown below. Leave default settings for all other fields.

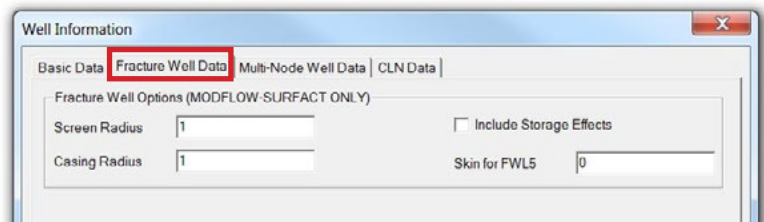
CLN Data tab: All fields should be left as default.

Editing an existing well

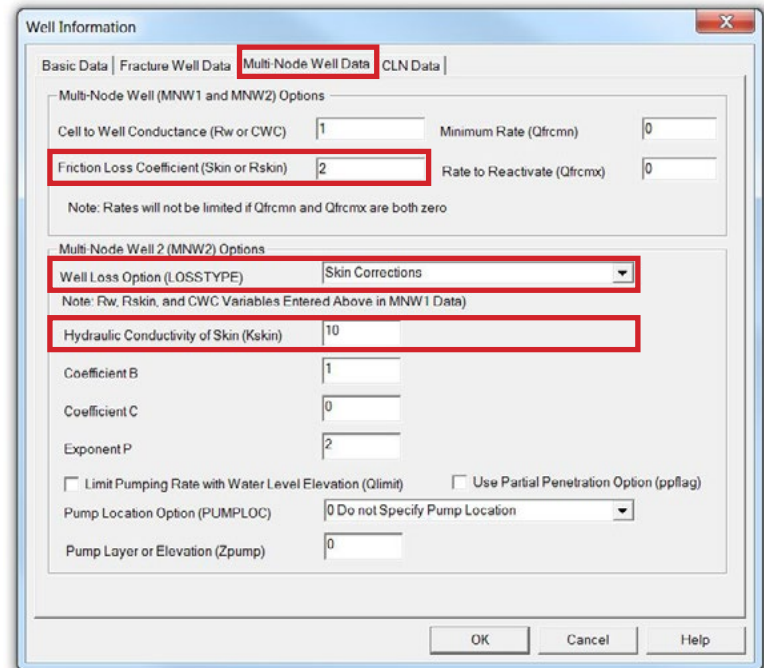
To edit an existing well, such as increasing the pumping rate, make the changes to the Basic Data tab. Refer to the “Adding a New Well” section for instructions on accessing the Basic Data tab.

Removing an existing well

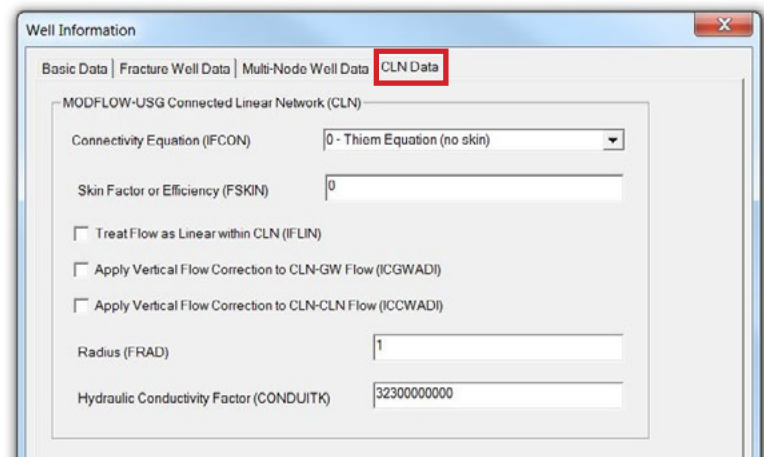
To remove existing wells, they first need to be selected using Edit > Select By or Edit > Select All. Then press the delete key to remove the wells. You can confirm that wells were removed by checking the total number of wells listed on the model information window (Model > Model Information). If you 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.



Adding a new well—Fracture Well Data tab settings (no changes)



Adding a new well—Multi-Node Well Data tab settings



Adding a new well—CLN Data tab settings (no changes)



Groundwater contribution to wells by layer (borehole flow)

A benefit of using the Multi-Node Well package is the ability to obtain a detailed record of the groundwater contribution of each model layer to the overall groundwater withdrawal from a particular well. For example, a well with an open interval from layers 9 to 12, would withdraw water from each of the four layers at a different rate depending on the hydraulic properties of the aquifer in each layer the well intersects. This allows you to determine which model layers are producing more or less water and may guide decisions regarding the optimum open interval of a well. The MNW package may also be used to determine passive groundwater flow between layers in unpumped wells. This is particularly applicable for multi-aquifer wells whose open interval spans the Eau Claire aquitard, allowing groundwater to directly move from the overlying aquifer system into the Mount Simon aquifer.

To look up the mass balance of groundwater flow within a well, see the .wel file created for each model run. This file includes the flow for each well by layer.

Springs

The model simulates major Dane County springs (Pheasant Branch, Culver, Donald, etc.) using the Streamflow Routing package and by inserting high-hydraulic conductivity nodes at and below the appropriate spring locations. Flow from springs can be determined by onscreen query as described in "Understanding Streamflow Reporting."

For a complete listing of model nodes that correspond to springs, refer to appendix 7B of the Dane County model report.

Recharge

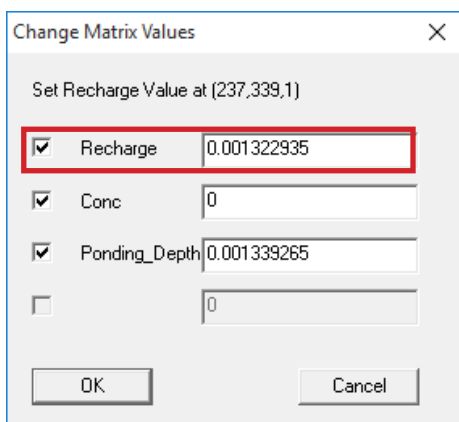
Groundwater recharge in Dane County varies spatially across the landscape. Initial values of recharge were taken from a soil-water-balance (SWB) model developed by Hart and others (2012), and were adjusted regionally during model calibration. In this model, evapotranspiration is accounted for in the recharge values so that recharge represents the water that actually reaches the water table.

Recharge can be viewed by selecting Props > Recharge. The lower-left corner displays recharge in units of feet/day. Ponding depth is also shown; the model allows ponding up to this depth if the water table rises above land surface.

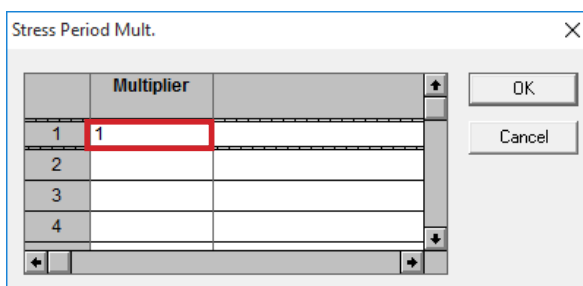
Users may wish to modify the recharge to simulate the steady-state conditions following land development or a change in climate. However, recharge should be modified with caution due to the complexity of the initial value distribution and the unknown consequences of future climate change (such as increased 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.

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.



Setting recharge for individual cells



Use multipliers to make global changes to recharge



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.

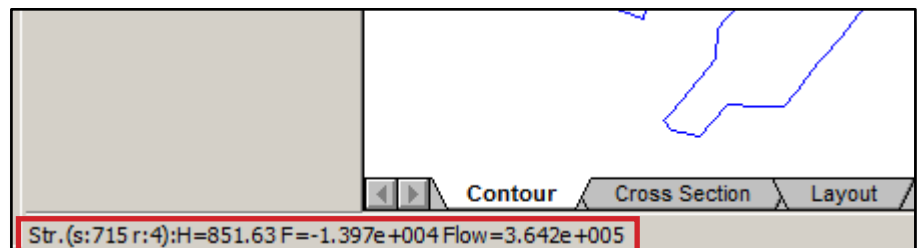
Following a run, once model results (including the cell-by-cell flow .cbb file) have been imported, Vistas can be queried to show the simulated streamflow at any stream point. To do this, you must turn on the Stream Boundary Condition (BCs > Stream) or specify Stream in the drop-down Boundary Condition 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 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 below, 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 feet per day (*again, negative flux means water is moving out of the groundwater system and into the stream*)
- Flow = 3.642×10^5 cubic feet per 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. In contrast, SFR2 segments that receive additional contributions of water from wastewater treatment plant discharges, would record a jump in total cumulative flow. For more discussion about wastewater treatment plant discharges refer to the “Streamflow Routing” section.



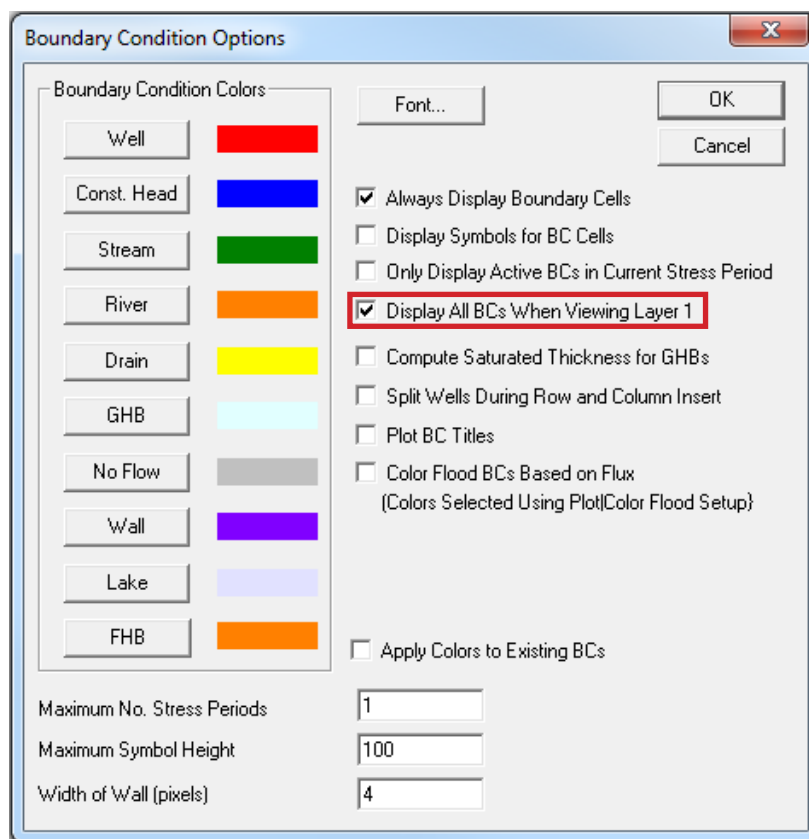
Sample streamflow report



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 compiling streamflow information and comparing one run to another. This macro, the ArcMap SFR2 Baseflow Comparison macro, is documented at the end of this guide.



Uncheck box to view SFR2 cells present in layer 1 only



Particle tracking using MODPATH

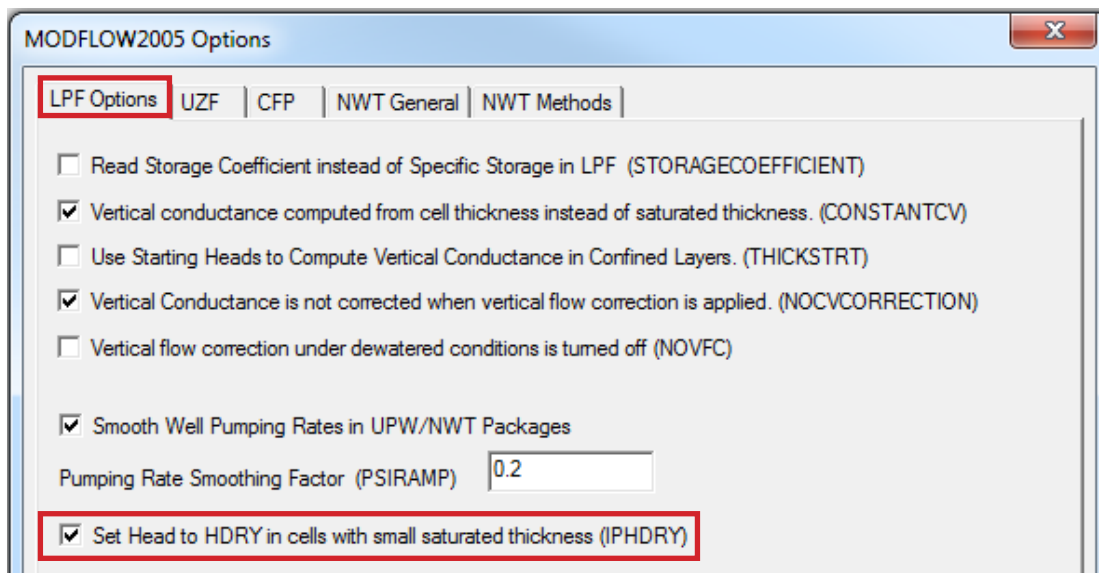
Particle tracking is a powerful way to help visualize the direction of groundwater flow and evaluate the impact of new or existing wells, simulate flow away from a landfill, or towards a subdivision. Particles can be simulated in both forward and reverse particle tracking mode, allowing the user to investigate 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.

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 you plan to compare different model runs to calculate changes in drawdown, be aware that you won’t be able to use these heads from MODFLOW as starting heads because many of the cells will start out dry. To get around this, see the section “Calculating Drawdown”.



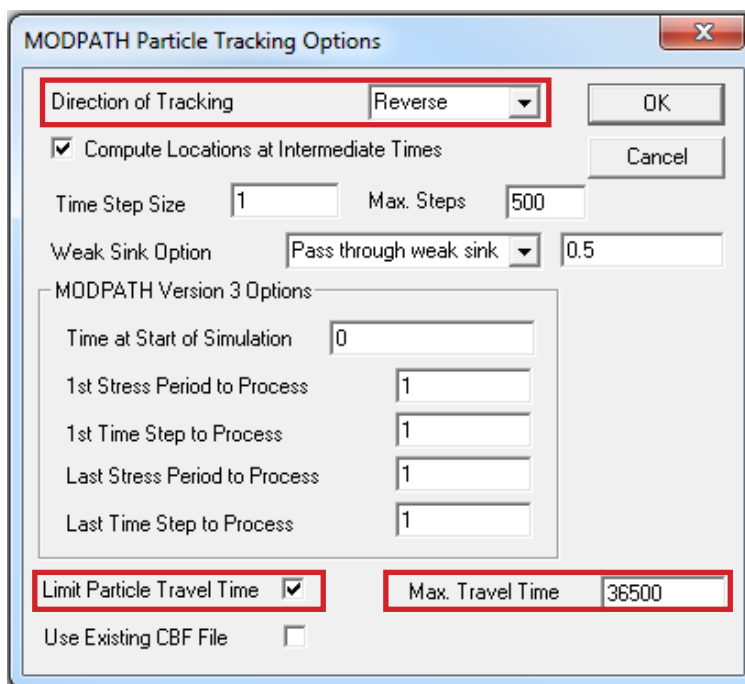
Check box to keep dry cells from causing MODPATH to crash



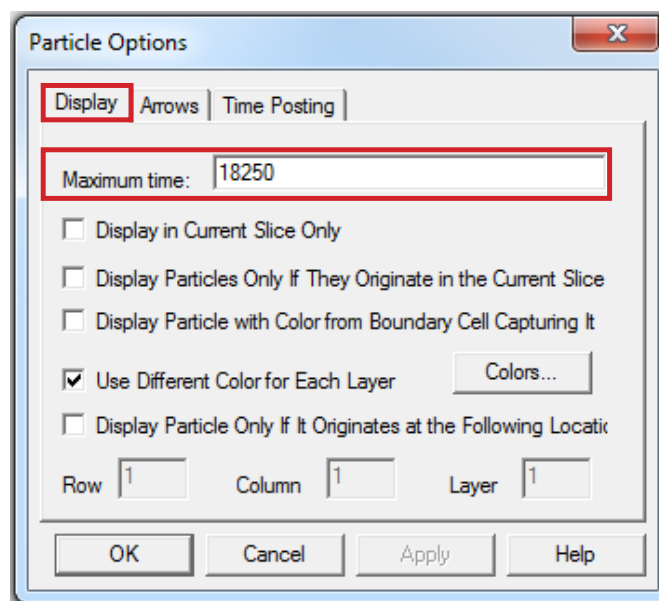
The following example demonstrates how to perform a capture zone (area of contribution) analysis for a well by delineating a 100-year time-of-travel area.

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 Time" and entering the "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 the "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.



Settings for particle tracking



Displayed maximum travel time can be adjusted here if viewing particle traces in Vistas only (not exporting)



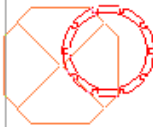
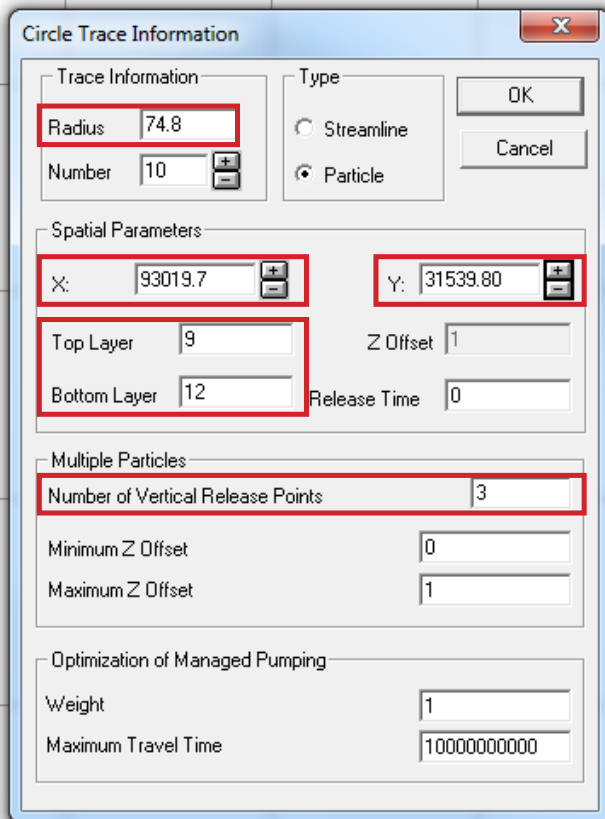
Once you have set the particle options in Model > MODPATH > Particle Options..., the next step is to insert circles of particles for the well of interest. Under the Analytic Element (AE) drop-down window, select AE > Particle > Circle. When you click on “Circle” the window will disappear and you can 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 the Circle Trace Information (see screenshot).

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.

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 180 ft since the cell size equals 360 x 360 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 74.8 ft. The radius of the particle circle should be 74.8-ft based on the

NOTE: The effective radius for the near-field of the Dane County model is calculated as 360 ft (model grid interval) ÷ 4.81 = 74.8 ft.

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



size of the model cells (360 x 360 ft) and the principles of converging flow near a pumped well node (Prickett and Lonquist, 1971).

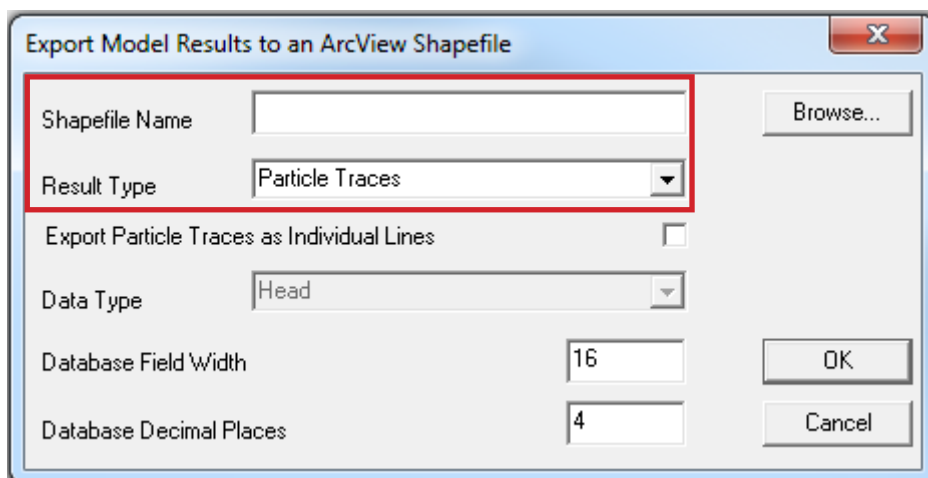
When doing reverse particle tracking to delineate capture zones for wells, we recommend using three circles of 10 particles in each open interval layer of the well. In other words, if the well is a multi-aquifer well which is open between model layers 9 and 12 you would place three circles of 10 particles in layers 9, 10, 11, and 12 (a total of 12 circles and 120 particles). You will also need to specify the open interval of the well ("Top Layer" and "Bottom Layer") and the "Number of Vertical Release Points" to "3". For this example, set the top layer to 9 and the bottom layer to 12. 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.

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.

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).



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 2010 conditions, or evaluating the impact of adding or removing new wells. Regardless of the objective, it is important to follow these steps:

First, save and run a version of the model that reflects initial drawdown conditions. For example, if interested in determining drawdown in 2010 compared to predevelopment, then predevelopment drawdown (no wells or wastewater treatment plant discharges) represents initial conditions. If interested in determining drawdown under an alternative scenario (say, wells removed or wells added), then 2010 drawdown represents initial conditions. Refer to the “Wells” and “Streamflow Routing” sections 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, 2010, 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. Same as for the initial-conditions version, the box “Set Head to HDRY in cells with small saturated thickness (IPHDRY)” needs to be unchecked before running the model. 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

There are two Dane County transient models, the new well example and the calibration example. Both are described in additional detail in the modeling report. The new well example demonstrates the transient response of a new high-capacity well pumping over a 30-day period, and is typical of the type of analysis commonly needed for new high-capacity wells. The simulation uses two stress periods (30 and 60 days), each with 10 time steps, for a total simulation time of 90 days. This example is set up to calculate streamflows at two model nodes using the MODFLOW GAGE package (see “Using GAGE Package for Streams or Lake Stage”).

The calibrated Dane County transient model simulates drought conditions during June and July 2012. The model is divided into four stress periods of 10 to 14 time steps each to represent periods of different pumping rates. Each time step is equivalent to one day, for a total simulation time of 49 days. Recharge is set to zero throughout the simulation period.

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

We have not had success running the transient model using just the datasets created by Groundwater Vistas, but have provided the modified files necessary to run the transient model. Transient simulations require some initial file setup (see “Technical Setup”) and must be run slightly differently than steady-state simulations (see “Running the Transient Model”).

Best practices for successful transient runs

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

- Use initial heads and lake levels as near to the final solution as possible. This should not be an issue unless 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 recharge, streamflow routing, lakes, boundary conditions, and well pumping. See “Editing Transient Model Parameters” for details.
- Run Vistas using the native executable file and correct file settings. This includes the NWT package, lake

bathymetry, and name (.nam) files as described in “Files Needed to Run the Transient Model.”

- Avoid increasing the duration of time steps. The model lakes are particularly unstable for large time steps.
- 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 Streams or Lake Stage” for more details.)

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.

Files needed for running the transient model

The transient model should be run using the native USGS executable (MODFLOW-NWT) as it is the “official” and most up-to-date USGS solver. This is the same executable file described for steady state in “Running the Model,” available for download on the USGS Groundwater Information page: water.usgs.gov/ogw/modflow-nwt/. This file is typically saved with other computer program files.

A number of files provided by WGNHS are also necessary to run the transient model. The following files should be placed in the working directory (as set in Model > Paths to models). Most of these files should not need to be changed. However, the name (.nam) file must be changed any time the Groundwater Vistas root file name is changed, as described in “Changing the Name File”. The lake files need not be changed unless lake properties are changed. See MODFLOW documentation for further description of the files and their format (water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/).



Provided files needed to run the transient model

Details	File name
Bathymetry —gives stage/ volume relation for each lake	lak_bath1.txt (Mendota)
	lak_bath2.txt (Wingra)
	lak_bath3.txt (Monona)
	lak_bath4.txt (Waubesa)
	lak_bath5.txt (Kegonsa)
	lak_bath6_Jul13.txt (Fish)
	lak_bath7_Jul13.txt (Crystal)
Lake-bed leakance	moddane5_BDLKNC.dat
Newton (NWT) solver package	Dane9_rjh_4Mar14.nwt
Lake package	Moddane7-5_rgn_mar14_trans.lak (calibrated model)
	Moddane7-5_rgn_mar14_trans_for_dd_test.lak (new well model)
Name file —specifies names of input and output files	Dane_trans_final_USETHIS.nam (calibrated model)
	Transient_new_well_example_USETHIS.nam (new well model)

See the above list for names of provided files needed to run the transient model.

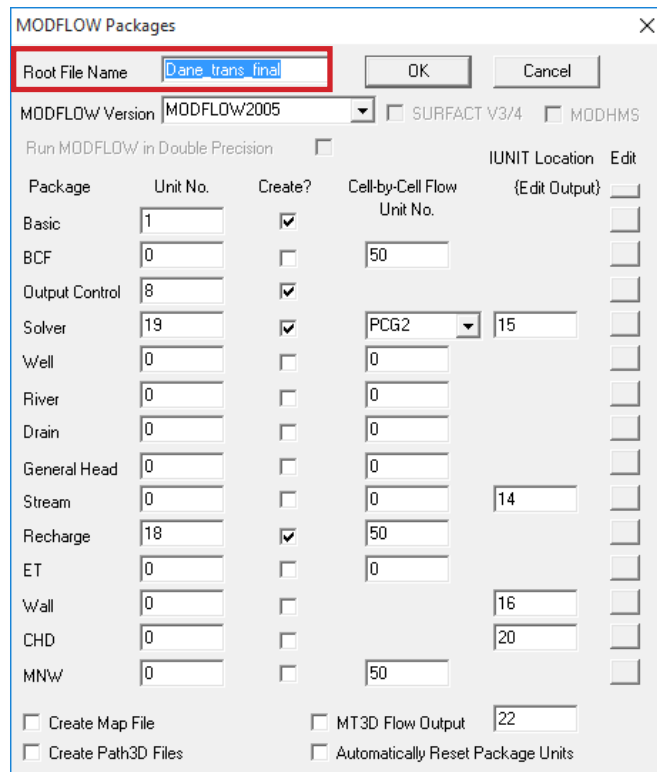
NOTE: The name (.nam) file must be changed whenever the Groundwater Vistas root file name is changed. See “Changing the Name File” for details.

Changing the name file

When you create datasets in Vistas, many files are generated, including a name (.nam) file. This file is a list of all the input and output files MODFLOW needs to run the model and so is critical for the run. However, we recommend using modified versions of some of the input/output files, which have been provided with the model

(see “Files Needed for Running the Transient Model Outside of Vistas”). The name file must reference these modified input/output file names.

Note: We have provided an example name file for each transient model. You may use this file for running the transient model unless you change the root file name. The root file name can be found in Model > MODFLOW > Packages (see screenshot). If you change the root file name, you will need to create your own edited name file. The name file is used later in “Running the Transient Model.”



Specify the root file name



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 replace two lines of text with the appropriate edited files. Because these file entries are slightly different for the two transient models, a readme file with instructions has been included with each model.

The new name file should look similar to the screenshot (below right), except “Dane_trans_final” will be replaced by your root file name. The yellow highlight shows the edited 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.

BEFORE

```
Dane_trans_final.nam - Notepad
File Edit Format View Help
LIST 7 Dane_trans_final.lst
BAS6 1 Dane_trans_final.bas
DIS 29 Dane_trans_final.dis
UPW 11 Dane_trans_final.upw
ZONE 40 Dane_trans_final.zone
MNW2 23 Dane_trans_final.mnw2
MNWI 45 Dane_trans_final.mnwi
DATA 90 Dane_trans_final_mnw2.wel
DATA 85 Dane_trans_final.byn
DATA 86 Dane_trans_final.qsm
SFR 27 Dane_trans_final.sfr
RCH 18 Dane_trans_final.rch
OC 8 Dane_trans_final.oc
NWT 19 Dane_trans_final.nwt
LAK 35 Dane_trans_final.lak
DATA(BINARY) 50 Dane_trans_final.cbb
DATA 73 Dane_trans_final1._kx
DATA 74 Dane_trans_final2._kx
DATA 75 Dane_trans_final3._kx
DATA 76 Dane_trans_final4._kx
DATA 77 Dane_trans_final5._kx
DATA 78 Dane_trans_final6._kx
DATA 79 Dane_trans_final7._kx
DATA 80 Dane_trans_final8._kx
DATA 81 Dane_trans_final9._kx
DATA 82 Dane_trans_final10._kx
DATA 83 Dane_trans_final11._kx
DATA 84 Dane_trans_final12._kx
DATA 70 Dane_trans_final._kz
DATA(BINARY) 30 Dane_trans_final.hds
DATA(BINARY) 31 Dane_trans_final.ddn
```

Use a text editor to replace highlighted lines as shown; refer to the readme file included with the model for line entries

AFTER

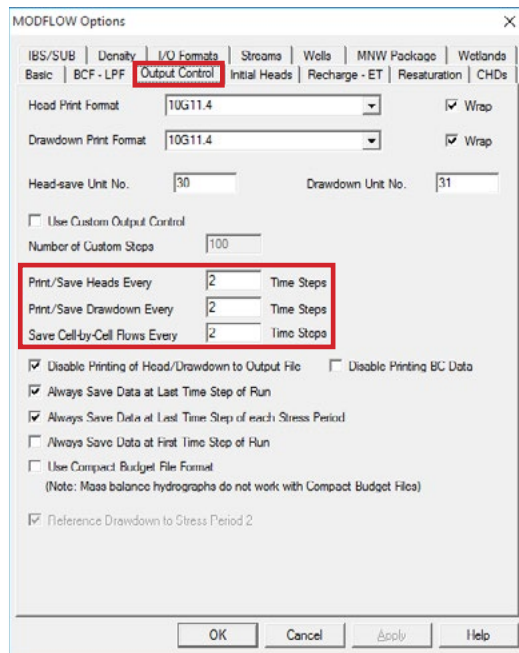
```
Dane_trans_final_USETHIS.nam - Notepad
File Edit Format View Help
LIST 7 Dane_trans_final.lst
BAS6 1 Dane_trans_final.bas
DIS 29 Dane_trans_final.dis
UPW 11 Dane_trans_final.upw
ZONE 40 Dane_trans_final.zone
MNW2 23 Dane_trans_final.mnw2
MNWI 45 Dane_trans_final.mnwi
DATA 90 Dane_trans_final_mnw2.wel
DATA 85 Dane_trans_final.byn
DATA 86 Dane_trans_final.qsm
SFR 27 Dane_trans_final.sfr
RCH 18 Dane_trans_final.rch
OC 8 Dane_trans_final.oc
NWT 19 Dane9_rjh_4Mar14.nwt
LAK 35 Moddane7-5_rgn_mar14_trans.lak
DATA 60 lak_bath1.txt
DATA 61 lak_bath2.txt
DATA 62 lak_bath3.txt
DATA 63 lak_bath4.txt
DATA 64 lak_bath5.txt
DATA 65 lak_bath6_Jul13.txt
DATA 66 lak_bath7_Jul13.txt
DATA 22 moddane5_BDLKNC.dat
DATA(BINARY) 50 Dane_trans_final.cbb
DATA 73 Dane_trans_final1._kx
DATA 74 Dane_trans_final2._kx
DATA 75 Dane_trans_final3._kx
DATA 76 Dane_trans_final4._kx
DATA 77 Dane_trans_final5._kx
DATA 78 Dane_trans_final6._kx
DATA 79 Dane_trans_final7._kx
DATA 80 Dane_trans_final8._kx
DATA 81 Dane_trans_final9._kx
DATA 82 Dane_trans_final10._kx
DATA 83 Dane_trans_final11._kx
DATA 84 Dane_trans_final12._kx
DATA 70 Dane_trans_final._kz
DATA(BINARY) 30 Dane_trans_final.hds
DATA(BINARY) 31 Dane_trans_final.ddn
```



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 transient calibration run has 49 time steps; saving all the data for every time step generates a cell-by-cell (.cbb) file of almost 4 gigabytes.

The model is currently set to save every two time steps (equivalent to two days). This is set in Model > MODFLOW > Package options > Output control tab (see screenshot).



Set time steps

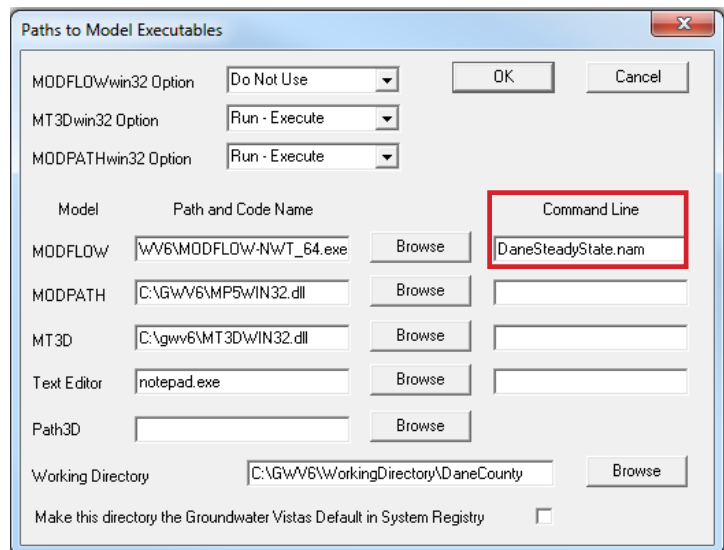
Running the transient model

As with the steady-state model, the transient model can be run in several ways. The most important difference is that the modified name file (see “Changing the name file”) must be referenced.

By running the USGS executable file in Vistas

The transient model may be run using the .exe method similar to how the steady-state would be run (see “Running the Model”). However, the edited name file **must** be specified (see “Changing the Name File”). In Models > Paths to models, enter the edited name file that you have saved in the command line. It is not case-sensitive. Your screen should look like the screenshot shown below.

Now you can run the model as you would for the steady-state. See “Monitoring a Transient Run” for details on viewing model progress.



Running the USGS executable file in Vistas—specify the name file in the command line



By clicking on the executable (.exe) file

You may also run the model 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 modified name file in the DOS command line. The model will begin running as described in the previous section.

By using a Windows batch file (.bat)

A third 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 Dane_trans_final_USETHIS.nam  
  
MODFLOW-NWT_64.exe Dane_trans_new well_USETHIS.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

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:

1. Look at the stress period and time step numbers in the DOS window. The numbers should gradually increase, changing every few minutes.
2. Look at the size of the list (.lst) file; it should gradually increase while the model is running.
3. 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

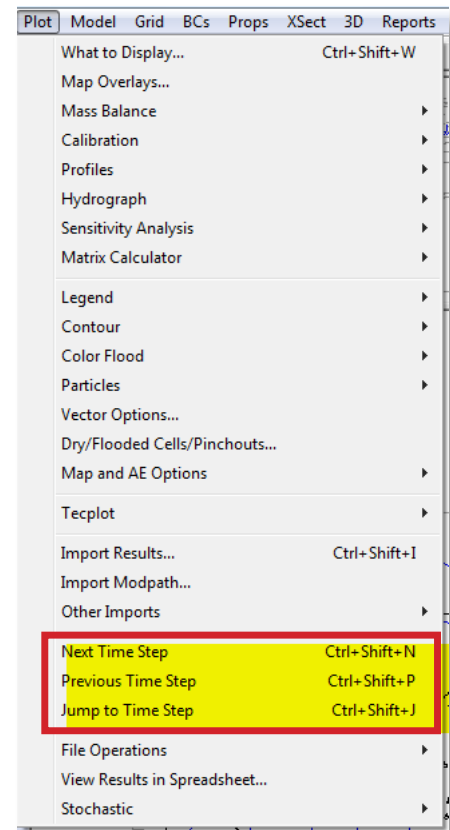
Transient output files (heads, drawdowns, mass balance) can be imported into Groundwater Vistas similar to steady-state runs, with the difference that you must specify the stress period and time step desired for viewing. Be aware that import might be slow because the output files can be very large, and Vistas has to sort through the large files to load the correct time step. This model has four stress periods with up to 14 time steps (days) 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).

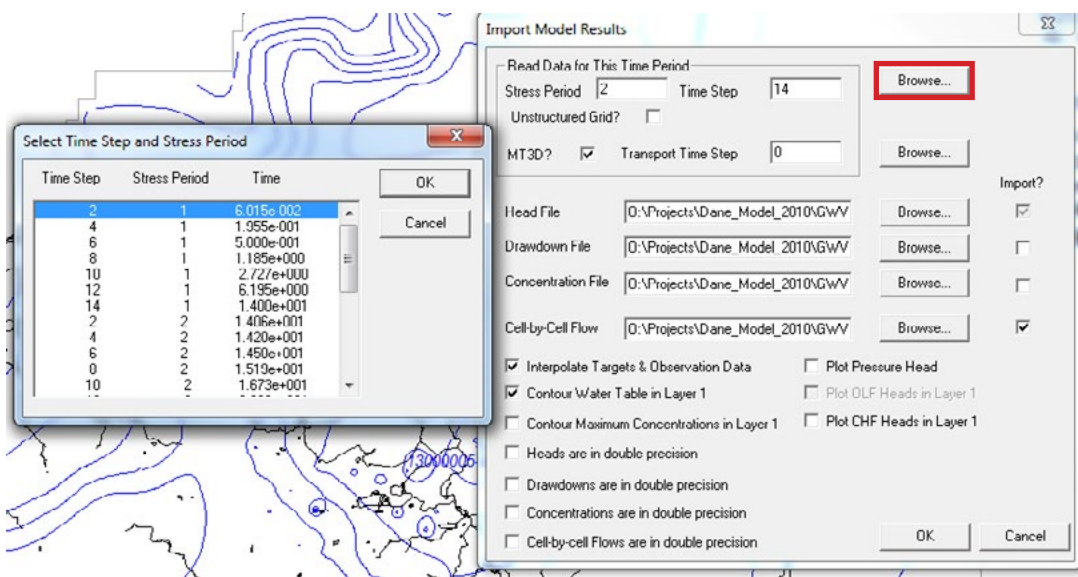
Note that changing the stress period on the left sidebar only changes the displayed input values, not results.

Changing the stress period in the sidebar changes displayed input values only, not results

Row Number:	135
Column Number:	383
Layer Number:	1
Stress Period:	4
Component Number:	1
Figure Number:	1
Sub-Layer Number:	1



A handy shortcut for switching between time steps



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.

Target Information

Target Type: Head

Target Name: []

Steady-State Data
 Target is Steady-State

Target Value: 0

Weight: 1

Group No.: 1

Species No.: 1

Minimum Bound: 0

Maximum Bound: 0

Spatial Parameters

X: 1899899.48 Y: 987759.86

Layer: 1 Sublayer: 1

Head Target is Difference with Layer: 0

Censoring Option: 0 - No Censoring

Transient Data... Graph... Import...

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

Transient Calibration Target Information

Transient

Number of Observation Times: 5 Update

	Time	Target Value	Weight	
1	0	0	0	
2	0	0	0	
3	0	0	0	
4	0	0	0	
5	0	0	0	
6				
7				

Transient Data... OK Cancel Apply Help

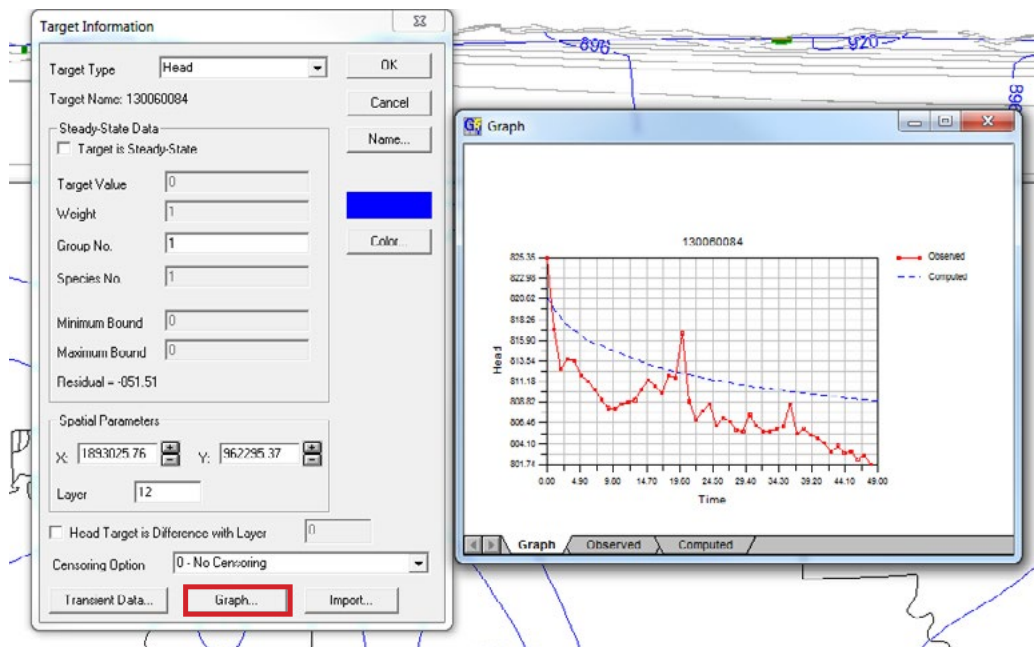
Transient data entry area



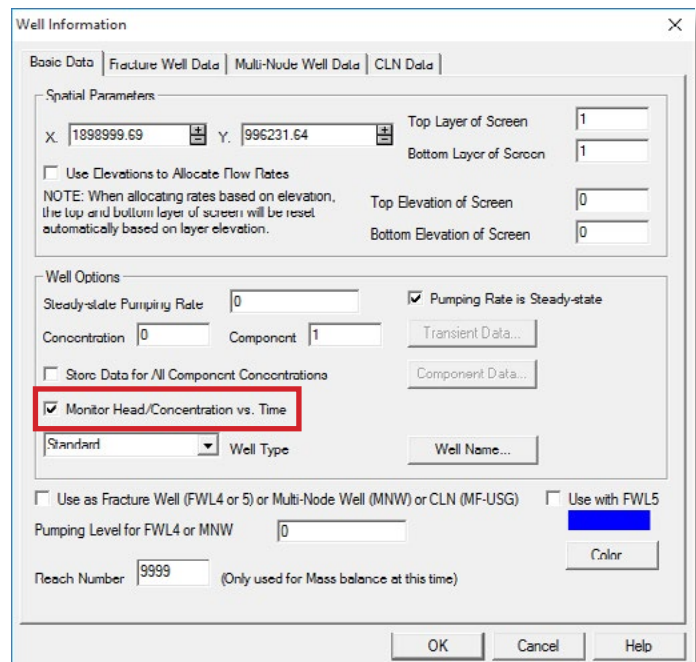
The 2016 Groundwater Flow Model for Dane County, Wisconsin—User’s Manual

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 streams or lake stage

We recommend using the GAGE package when plotting transient hydrographs of flows or lake stages. 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 or lake nodes. An example of this is included with the new well transient model.

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

TIP: We find it easier to prepare the gage file using a text editor rather than Vistas. Before setting up the file, read the MODFLOW documentation on the GAGE package: water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/gage.htm.

To use the GAGE package you must add the gage file and the specified output files to the MODFLOW name (.nam) file. The word “GAGE” must appear in the name file. Here is an example of the name file using the gage file above:

Explanation of highlighted text in the screenshot:

GAGE 67: References gage file (gagefile.dat) from the previous screenshot. “67” is a unit number not being used elsewhere. “GAGE” must be written out.

Data 51–53: Specifies the name of the output file for each gage and unit number (51–53) referenced in the gage file.

The screenshot below shows the partial contents of one of the gage output files. You can use another program (such as Grapher or Excel) to plot the hydrographs.

SAMPLE GAGE FILE

(see MODFLOW documentation for details)

segment	reach	output unit #	output flag
3			
250	1	51	1
250	6	52	1
866	4	53	1

```

LIST 7 new_well_test_trans.lst
BAS6 1 new_well_test_trans.bas
DIS 29 new_well_test_trans.dis
UPW 11 new_well_test_trans.upw
ZONE 40 new_well_test_trans.zone
MNW2 23 new_well_test_trans.mnw2
MNI 45 new_well_test_trans.mni
DATA 90 new_well_test_trans.mnw2.wel
DATA 85 new_well_test_trans.byw
DATA 86 new_well_test_trans.gsm
SFR 27 new_well_test_trans.sfr
RCH 18 new_well_test_trans.rch
OC 8 new_well_test_trans.oc
NWT 19 dane9_rjh_4mar14.nwt
LAK 35 MODDANE7-5_rgn_Mar14_trans_for_dd_test.lak
DATA(BINARY) 50 new_well_test_trans.cbb
DATA 73 new_well_test_trans1._kx
DATA 74 new_well_test_trans2._kx
DATA 75 new_well_test_trans3._kx
DATA 76 new_well_test_trans4._kx
DATA 77 new_well_test_trans5._kx
DATA 78 new_well_test_trans6._kx
DATA 79 new_well_test_trans7._kx
DATA 80 new_well_test_trans8._kx
DATA 81 new_well_test_trans9._kx
DATA 82 new_well_test_trans10._kx
DATA 83 new_well_test_trans11._kx
DATA 84 new_well_test_trans12._kx
DATA 70 new_well_test_trans._kz
DATA(BINARY) 30 new_well_test_trans.hds
DATA(BINARY) 31 new_well_test_trans.ddn
DATA 60 lak_bath1.txt
DATA 61 lak_bath2.txt
DATA 62 lak_bath3.txt
DATA 63 lak_bath4.txt
DATA 64 lak_bath5.txt
DATA 65 lak_bath6 Jul13.txt
DATA 66 lak_bath7 Jul13.txt
DATA 22 moddane5_BDLKNC.dat
GAGE 67 gagefile.dat
Data 51 Storyspring.dat
Data 52 HwyA.dat
Data 53 Bellbrook.dat

```

Add gage file and output file names to the MODFLOW name file

GAGE No.	Time	Stage	Coord.	Flow	Depth	width	Midpt-Flow	Precip.	FT	Runoff
2	6471350F-01	9.0260718F+02	1,410,285	4.1362506F+05	6.7215465F-02	2.4943256F+02	4.0801950F+05	0.0000000F+00	0.0000000F+00	0.0000000F+00
6	6178370E-01	9.0260712E+02	1,410,285	4.1273547E+05	6.7117788E-02	2.4943256E+02	4.0705175E+05	0.0000000E+00	0.0000000E+00	0.0000000E+00
1	2573891E+00	9.0260699E+02	1,410,285	4.0762625E+05	6.7005284E-02	2.4943256E+02	4.0591522E+05	0.0000000E+00	0.0000000E+00	0.0000000E+00
7	1507971E+00	9.0260651E+02	1,410,285	4.0681359E+05	6.6510104E-02	2.4943256E+02	4.0092791E+05	0.0000000E+00	0.0000000E+00	0.0000000E+00

Partial contents of a gage output file



Editing transient model parameters

Transient model properties must be set up for all stress periods. Properties with transient data include recharge, streamflow routing, lakes, 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. 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. 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 putting in the range of stress periods you wish to copy to. The same technique applies to streams and lakes.

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 four stress periods.

The screenshot shows the 'Well Information' dialog box with the following settings:

- Spatial Parameters:** X: 1866845.75, Y: 970638.19, Top Layer of Screen: 9, Bottom Layer of Screen: 12, Top Elevation of Screen: 0, Bottom Elevation of Screen: 0.
- Well Options:** Steady-state Pumping Rate: -19264.69, Concentration: 0, Component: 1, Well Type: Standard, Well Name: X77044.
- Transient Data:** The 'Pumping Rate is Steady-state' checkbox is unchecked, and the 'Transient Data...' button is highlighted with a red box.
- Other Options:** 'Use as Fracture Well (FWL4 or 5) or Multi-Node Well (MNW) or CLN (MF-USG)' is checked. 'Pumping Level for FWL4 or MNW' is 0. 'Reach Number' is 1.

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

	Starting Stress Period	Ending Stress Period	Head (Q for Wells)	Concentration	Flow (Stream Only)	Width (Stream Only)	
1	1	1	-19264.69	0	0	0	
2	2	2	-22050.36417	0	0	0	
3	3	3	-23982.61258	0	0	0	
4	4	4	-24774.39134	0	0	0	
5							
6							

Example data for a well pumped at different rates for each of four stress periods



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 across Dane County.

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

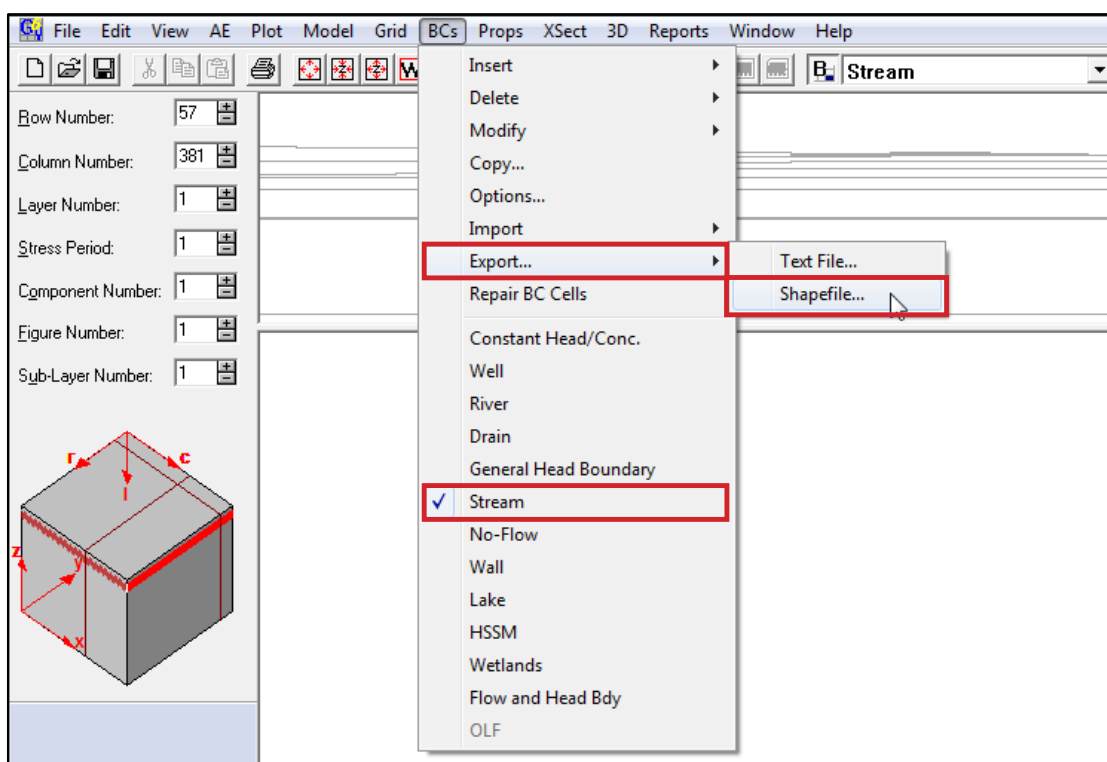
folder included with the model files and press Open. This will add the macro to the ArcToolbox for the open ArcMap session; you'll need to reinstall the macro each time you start a new ArcMap session.

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

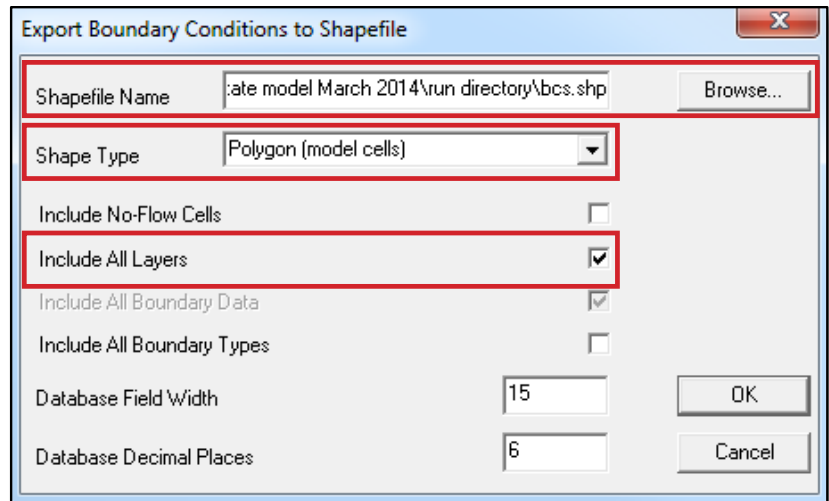
Under the tab BCs, highlight "Stream," then click Export > Shapefile (see screenshot).



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

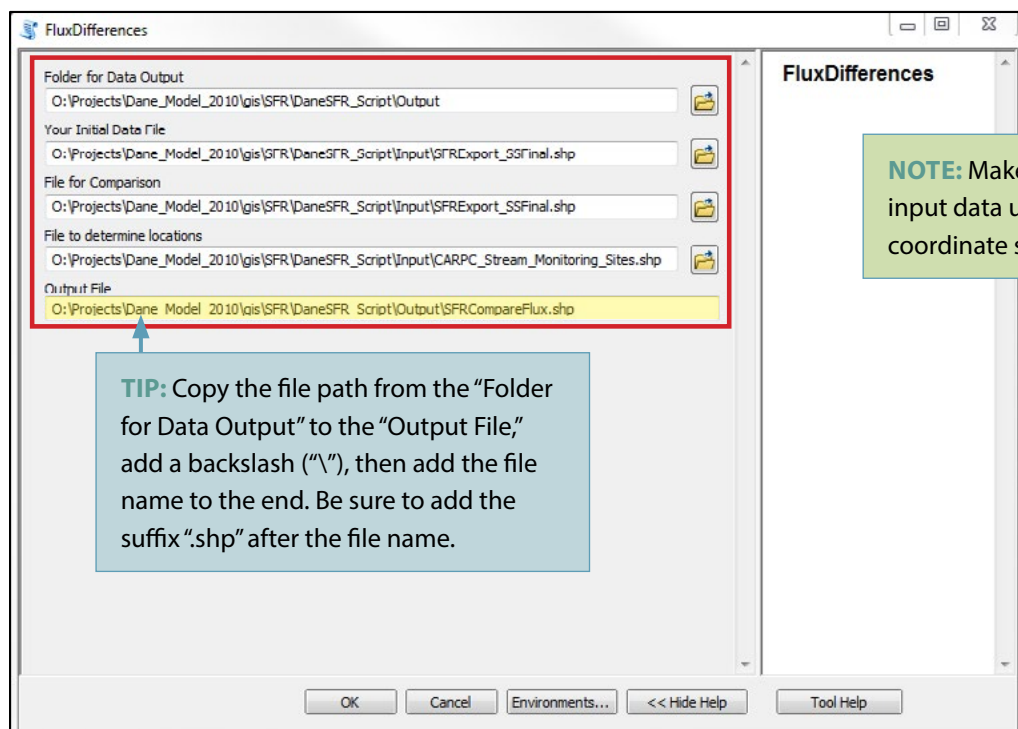
From the ArcToolbox window, open Flux Calculations and double click FluxDifferences to launch the macro (see screenshot).

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.



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

- 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., <http://pubs.usgs.gov/tm/2005/tm6A16/>.
- Krohelski, J.T., Bradbury, K.R., Hunt, R.J., and Swanson, S.K., 2000, Numerical simulation of groundwater flow in Dane County, Wisconsin: Wisconsin Geological and Natural History Survey Bulletin 98, 31 p.
- Parsen, M.J., Bradbury, K.R., Hunt, R.J., and Feinstein, D.T., in press, The 2016 Groundwater Flow Model for Dane County, Wisconsin: Wisconsin Geological and Natural History Survey Bulletin 110.
- Prickett, T.A., and Lonquist, C.G., 1971, Selected digital computer techniques for groundwater resource evaluation: Illinois State Water Survey Bulletin 55, 62 p.



Published by and available from:

Wisconsin Geological and Natural History Survey

3817 Mineral Point Road ■ Madison, Wisconsin 53705-5100
608.263.7389 ■ www.WisconsinGeologicalSurvey.org
Kenneth R. Bradbury, Director and State Geologist



This report is an interpretation of the data available at the time of preparation. Every reasonable effort has been made to ensure that this interpretation conforms to sound scientific principles; however, the report should not be used to guide site-specific decisions without verification. Proper use of the report is the sole responsibility of the user.

The use of company names in this document does not imply endorsement by the Wisconsin Geological and Natural History Survey.

ISSN: 0375-8265
ISBN: 978-0-88169-991-3

Issued in furtherance of Cooperative Extension work, Acts of May 8 and June 30, 1914, in cooperation with the U.S. Department of Agriculture, University of Wisconsin–Extension, Cooperative Extension. University of Wisconsin–Extension provides equal opportunities in employment and programming, including Title IX and ADA requirements. If you need this information in an alternative format, contact the Office of Equal Opportunity and Diversity Programs or the Wisconsin Geological and Natural History Survey (608.262.1705).

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.