

HYDROSIMULATICS, LLC
MLTISCALE ADAPTIVE GROUNDWATER NETWORK
(MAGNET)



MAGNET VERSION 3.0.0

USER'S MANUAL



Hydrosimulatics, LLC at Michigan State University



User's Manual for *MAGNET*

Copyright © 2018 by Hydrosimulatics, LLC at Michigan State University. All rights reserved.

Hydrosimulatics, LLC makes no warranties either expressed or implied regarding the program *MAGNET* and its fitness for any particular purpose, or for the validity of the information contained in this document.

Authors / Editors:

Zachary Curtis

Dr. Hua-Sheng Liao

Dr. Shuguang Li

DOCUMENT VERSION: 1.0.0

LAST REVISED: March 29, 2021

<http://www.magnet4water.net>

TABLE OF CONTENTS

TABLE OF FIGURES.....	6
Chapter 1 INTRODUCTION & ACCESS	9
1.1 Overview of MAGNET.....	9
1.2 Datasets	10
1.3 User Accounts.....	10
1.4 MAGNET Premium	17
Chapter 2 MAGNET MODELING ENVIRONMENT.....	19
2.1 Maps Display and Working Area.....	20
2.2 Status Bar	21
2.3 Map Display Tools.....	21
2.4 Simulation Tools.....	22
2.5 Model Creation and Visualization Tools.....	22
2.6 Conceptual Features Tools	23
2.7 Utilities and Analysis Tools.....	23
Chapter 3 MODEL CREATION AND PARAMETER OPTIONS	25
3.1 Delineating the Model Domain.....	25
3.2 Model Grid and Solver Settings.....	27
3.3 Aquifer Elevations	28
3.4 Hydraulic Conductivity	32
3.5 Recharge and Surface Seepage	32
3.6 Boundary Conditions & Hierarchical Modeling	34
3.7 Transient Model Settings & Initial Conditions	35
Chapter 4 CONCEPTUAL FEATURES: LINES, ZONES & WELLS	36
4.1 Line Features & Attributes	36
4.2 Zone Features & Attributes	43
4.3 Well Features & Attributes.....	57
Chapter 5 SIMULATION AND VISUALIZATION	64
5.1 Submitting a Model for Simulation.....	66
5.2 Display Options	69
5.3 Model Values at a Node	74
5.4 Cross-sections	75
5.5 3D Visualizations.....	78
5.6 Particle Tracking.....	80
Chapter 6 ANALYSIS & MODEL PERFORMANCE.....	84

6.1	Monitoring Well Time-Series and Breakthrough Curves.....	84
6.2	Water Balance Analyses.....	85
6.3	Model Results vs. Observations (Manual Calibration)	89
Chapter 7	UTILITIES & FILE FUNCTIONS	98
7.1	Saving and Loading Models	98
7.2	Publishing & Editing MAGNET Network Models.....	99
7.3	Background Images.....	105

TABLE OF FIGURES

Figure 1-1: Account login/sign-up interface.	11
Figure 1-2: Hydrosimulatics Sign Up Form.....	12
Figure 1-3: Options for describing user job title/type and affiliation.	12
Figure 1-4: Hydrosimulatics Account Edit Form.....	14
Figure 1-5: View Message Box.....	14
Figure 1-6: Activity Report interface.	15
Figure 1-7: MAGNET User Available Download List.	15
Figure 1-8: Example MAGNET model result file.....	16
Figure 1-9: Signing in to a ‘Guest’ account.	17
Figure 2-1: MAGNET Modeling Environment upon login.	20
Figure 2-2: Using the Map Display Tools to customize the display.	22
Figure 2-3: Default Parameters and Options menu.	24
Figure 3-1: Creating and adjusting the model domain using the ‘DomainRect’ and ‘DomainZone’ button. The blue in-set map highlights the ‘un-do’ button available after altering any node position.....	26
Figure 3-2: The Model Grid and Solver settings available in the Default Parameters and Options menu. Default values/settings are shown.	27
Figure 3-3: Options and parameters for assigning aquifer elevations in MAGNET. Default values/settings are shown.	29
Figure 3-4: Different possible settings for the aquifer bottom elevations.	31
Figure 3-5: Options and settings for parametrizing the hydraulic conductivity of the aquifer. Default values/setting are shown.	32
Figure 3-6: Options for applying recharge along the top of the model boundary. Default values/settings are shown.	33
Figure 3-7: Surface leakance parameter text box.	33
Figure 3-8: Options for applying boundary conditions for flow and/or solute transport modeling.	34
Figure 3-9: Example of the hierarchical modeling approach for developing appropriate boundary conditions in the study area.....	35
Figure 3-10: Options for unsteady (transient) modeling in MAGNET. Default values/parameters are shown.....	36
Figure 3-11: Options for assigning an initial head distribution across the model domain. Default value/parameters are shown.	36
Figure 4-1: Polyline feature added to the model domain and the Polyline Attributes menu.	37
Figure 4-2: Sub-menu for editing Transient prescribed head data.	38
Figure 4-3: Edit Polyline Attributes sub-menu for assigning unique head values at different vertex locations. Default values/options are shown.....	39
Figure 4-4: Edit Polyline Attributes sub-menu for assigning polyline attributes needed for computing two- way head dependent flux. Default values/options are shown.	40
Figure 4-5: Edit Polyline Attributes sub-menu for assigning polyline attributes needed for computing one- way head dependent flux. Default values/options are shown.	41
Figure 4-6: Sub-menu for editing Transient prescribed head data.	42
Figure 4-7: Editing attributes of a pre-existing polyline feature.	43
Figure 4-8: Zone feature added to the model domain and the Zone Attributes menu.	44
Figure 4-9: Elevation sub-menu (for Zone conceptual features).....	46

Figure 4-10: Example Cross-section results (see Section 5.4) of using and not using Partial Penetration for a zone feature in MAGNET. When partial penetration is not used, the solute source is distributed throughout the aquifer system (i.e., across all computational layers. When it is used, the solute source is restricted to the topmost computational layer.	47
Figure 4-11: Transport Property sub-menu (for Zone conceptual features).	48
Figure 4-12: Biochemical sub-menu (for Zone conceptual features).	49
Figure 4-13: Sources and Sinks (Head Dependent) sub-menu (for Zone conceptual features).	50
Figure 4-14: Sub-menu for editing transient lake (or other source/sink) stage data.....	50
Figure 4-15: Sub-menu for editing transient lake concentration data.	51
Figure 4-16: Sources and Sinks (Prescribed) sub-menu (for Zone conceptual features).....	52
Figure 4-17: Sub-menu for editing transient recharge data.	53
Figure 4-18: Sub-menu for editing transient recharge concentration data.....	54
Figure 4-19: Sub-menu for editing transient prescribed head data.....	55
Figure 4-20: Sub-menu for editing transient source concentration data.	56
Figure 4-21: attributes of pre-existing zone features.	57
Figure 4-22: Well Input Options menu.....	58
Figure 4-23: Sub-menu for editing transient pumping/injection rate data.	59
Figure 4-24: Sub-menu for editing transient injection source concentration data.....	60
Figure 4-25: Particle tracking options available in the Default Parameters and Options menu. Default values/settings are shown.....	60
Figure 4-26: Sub-menu for editing transient head observations.	62
Figure 4-27: Sub-menu for editing transient concentration observations.	62
Figure 4-28: Example of a well feature placed in a model.	63
Figure 5-1: Projection option box in the Default Parameters and Options menu.....	67
Figure 5-2: Example of simulation results (head and concentration). Note that the solute concentration is indicated with the color overlay (red = high concentration; blue = low concentration).....	68
Figure 5-3: Display options available in MAGNET.....	69
Figure 5-4: Different ways to display the groundwater flow results (head distribution and seepage velocities).....	70
Figure 5-5: Drop-down menu used to select an model input for display.....	71
Figure 5-6: Examples of displaying model inputs in the MDWA.....	71
Figure 5-7: Example of using the 'Seepage Area' option from the 'Display Input' drop-down menu to indicate areas where groundwater is discharging at the land surface.	72
Figure 5-8: Example of displaying flow model results in Google Earth.	73
Figure 5-9: Example of using the 'HideOverlay' button to temporarily remove the simulation results from the MDWA.	74
Figure 5-10: Example of placing nodal value flags in the model domain, with a sample variable table shown for the flag in the upper right of the model domain.....	75
Figure 5-11: Example charts generated when using the 'Display Charts'. An example Cross-section plot is shown in the top-right, and an example Cross-section diagram (Full cross-section) is shown in the bottom-right.	76
Figure 5-12: Different configurations of the Cross-section plot for the Cross-section shown in Figure 5-11.....	77
Figure 5-13: Example Cross-section Diagram. Default options/settings are shown.	78
Figure 5-14: Customized Cross-section Diagram. The hydraulic conductivity (K) field is shown, with red colors indicating high K and bluer colors indicating low K.	78
Figure 5-15: Example 3D visualization of a model in MAGNET.	79
Figure 5-16: Model variables available for display in the 3D Surface Plot.....	80

Figure 5-17: Particle Pop-up menu.....	81
Figure 5-18: Example of placing particles along a polyline and performing forward particle tracking.....	82
Figure 5-19: Example of placing particles in a zone and performing reverse/backward particle tracking.	83
Figure 6-1: Example of using a monitoring well to generate a concentration breakthrough curve.....	85
Figure 6-2: Example of using a monitoring well to visualize time-series simulated head data.	85
Figure 6-3: Example Zone (water) Balance charts. Top-right:: zone budget for entire domain polygon; bottom-left: zone budget for a zone added to the model domain; bottom-right: zone budget for a single model cell.....	88
Figure 6-4: The 45 Degree Line Based Calibration Chart used for “manual calibration” (comparing current model outputs with observations).....	90
Figure 6-5: Calibration Data window, ready for direct input of observational data.....	91
Figure 6-6: Example header line and line of observation inputs.....	91
Figure 6-7: Calibration Data window after importing observation data from a file.....	91
Figure 6-8: Filter for Well Data window for importing data from the IGW server	92
Figure 6-9: Modeling Samples vs. Calibration plot, extracted model values and residuals (relative to the observations), and chart options.....	93
Figure 6-10: Scheme used for 3D Weighted interpolation of simulated head for comparison with observations.	94
Figure 6-11: Example Calibration plot with Confidence intervals (red dotted line) of one standard deviation and Band-mean indicators (yellow triangles).	96
Figure 6-12: Examples of residual mapping display options.....	97
Figure 7-1: Utilities sub-menu.	98
Figure 7-2: Example MAGNET model file.	99
Figure 7-3: Example of publishing a model to the MAGNET network using the Model Publishing Options window.	101
Figure 7-4: MAGNET Model Network, available at www.magnet4water.com/modelnetwork/	102
Figure 7-5: Selected model details in the MAGNET Model Network web interface.	103
Figure 7-6: Model Results and Comments page.	104
Figure 7-7: Window for editing user published models.	105
Figure 7-8: 'Overlay an Image on Maps' menu.	106

Chapter 1 INTRODUCTION & ACCESS

Understanding groundwater systems has traditionally been hampered by a lack of data - both for the analysis or characterization of the system and for conceptualizing, building, calibrating and validating numerical models. In recent times, though, the geospatial and digital revolutions have led to the availability of massive datasets such as high-resolution Digital Elevation Models (DEMs) or detailed climate information (precipitation, temperature). In spite of the availability of this “big data”, its size renders it immovable and hard to maneuver. As a result, the tools and techniques required to efficiently utilize this data have not kept pace with the rate of data “creation.” Moreover, the time required for processing the data makes it highly impractical to accomplish, especially considering that the same process must be repeated for every project. Obviously, this is an impractical way of efficiently using this data and needs improvement.

1.1 Overview of MAGNET

MAGNET is a data-enabled modeling system built on pre-processed, fully-assembled framework data, i.e., data needed to conceptualize and characterize the system (e.g., DEM, recharge estimates, etc.) in a ready-to-use form at the “global scale”. Even though the data provided by the system may not completely satisfy the requirements of a particular study, it provides the vast majority of data needed to build a preliminary model. Thus, the modeler can use the “data-ready” system to refine or modify the preliminary model as desired by adding site-specific details, and also guide further data collection efforts.

The system allows users not only to access and utilize pre-processed big datasets, but also to do so in real-time and interactively. Within the GIS-enabled modeling environment, users can zoom in anywhere to create a *data-enabled* model groundwater flow and contaminant transport, and immediately evaluate model performance using physical groundwater data available within the study area. All model inputs and representations (e.g. conceptual/structural, sources and sinks, number of layers, grid resolution, and display parameters) can be interactively modified and further customized based on study objectives and other available (local) data, as well as user experience and expertise. The users can: 1) change assumptions, conceptual framework, resolution, or representation, and immediately see the integrated effects, the ultimate implications, and the complex interrelationships presented intelligently in a meaningful context; 2) initiate or pause contaminant particle tracking or reactive plume modeling; and 3) perform well (head and concentration) monitoring and perform mass balance analyses.

The MAGNET modeling environment is designed in such a way that modeler feels as though they are using a single model that provides multiple options. The process starts with a global Google Map that allows users to select an area of interest, after which a geo-referencing model is determined based on the area selected. A GIS-based interface is provided for the user to fine-tune the computational domain and to add additional data such as i) polyline or polygon data used to represent a stream/lake in the model, or ii) geo-referenced images used as a base-map for digitizing a geological formation. For example, a polyline layer could be either treated as a stream with attributes of stage, leakance and bed elevation or as an injection trench with a prescribed flux; and a polygon layer could be defined as a lake with attributes of lake level and leakance, or as a contaminant source.

The remaining chapters of this User Manual explain the different functionalities of the MAGNET modeling platform. Chapter 2 overviews the MAGNET Modeling Environment and the assortment of modeling, visualization and analytical tools that are available. Chapters 3 through 6 guide users through the different

stages of model development and application – from model creation and parametrization, to adding conceptual features, running the simulation, and visualizing and analyzing the model results. Chapter 7 details the file function and model sharing utilities available in MAGNET.

1.2 Datasets

Table 1 provides a list of datasets that are currently enabled on the modeling system.

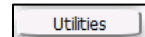
PARAMETER	RESOLUTION/Data Points	COVERAGE	USAGE
DEM	90 m	Global	Modeling
	30 m	USA	
	10 m	USA	
	3 m	USA*	
	1 m	USA*	
Rock Top Elevation	500 m	Michigan	Modeling
Surficial Aquifer Thickness	--	Michigan, Global ⁺	Modeling
Conductivity	500 m	Michigan, Global ⁺	Modeling
Recharge	1 mi	Michigan	Modeling/Calibration/Analysis
Static Water Levels	~500 million	Michigan	Calibration/Analysis
Soil Type	60 m	USA	Mapping
Land Use/Land Cover	30 m	USA	Mapping

1.3 User Accounts

All users of MAGNET must register an account, including a user name and user profile, at the Hydrosimulatics INC website:

<https://www.magnet4water.com/magnet/>

A palette of buttons is available along the left-side of the web interface (see Figure 2-1 in next Chapter). Click the ‘Utilities’ button, then click ‘My Account’ in the submenu that appears. This launches the Account Login/Sign-up Interface (see Figure 1-1).



Users may access the login/registration submenu at: <https://www.magnet4water.com/> (click the ‘Sign up’ link’).

×

You need to login first before you can implement this operation:

User Name

Enter User Name

Password

Enter Password

Master Account [Forget Password?](#)

Login

New user? please sign up here

Figure 1-1: Account login/sign-up interface.

New users should click the green ‘Sign Up’ button to create a user name and profile with the Hydrosimulatics User Sign Up Form (see Figure 1-2). Specify a username that will be provided when submitting a job to the MAGNET server (see Section 5.1) and a valid email account. Also specify a password that is needed when logging in with the provided username.

Users need to provide a basic description of their job title/type and affiliation. This is done with the two drop-down menus in the User Sign Up Form. The list of different job types and different types of affiliations are shown in Figure 1-3.

Hydrosimulatics Sign Up Form

Please fill in this form to create an account.

User Name
NOTE:: At least 3 characters and the first character must be a letter; the second and third must be letter or number

Enter User Name

Email

Enter Email

Password

Enter Password

Repeat Password

Repeat Password

Please select one that best describes you:

Please select one that best describes your affiliation:

GuestPassword

Enter Password for Guests

Additional Information

Enter other information

Use email notification for comment/message alert

By creating an account you agree to our [Terms & Privacy](#). **Accept**

Cancel
Sign Up

Figure 1-2: Hydrosimulatics Sign Up Form.

<ul style="list-style-type: none"> Undergraduate student <li style="background-color: #0070C0; color: white;">Graduate student Hydrologic modeler Hydrogeologist Hydrologist Water resources engineer Geotechnical engineer Geologist Geochemist Environmental physicist Environmental chemist Environmental biologist Environmental economist Environmental toxicologist Environmental scientist Environmental engineer Environmental manager/planner Someone who are concerned about the environment Other 	<ul style="list-style-type: none"> <li style="background-color: #0070C0; color: white;">University Other educational institutions National//State//Provincial Research Lab Consulting firm Corporate business State//provincial government agency Federal government agency Local city//county government agency NGO Nonprofit organization Other
---	--

Figure 1-3: Options for describing user job title/type and affiliation.

The Guest Password can be used to allow others MAGNET users to load and view model results associated with the ‘Master’ user account, although guests cannot edit/refine the model (e.g., they cannot add conceptual features, change inputs, etc. (see more details in next subsection). However, users can publish their models for other users to view and build upon (see section 7.2).

Finally, users may add additional information (optional) and choose if they would like to receive, via email, i) notifications about product updates from Hydrosimulatics; and ii) comments about models published by the user (see Section 7.2). The default setting is to receive notifications/comments via email.

After completing all required fields, the user must read and agree to the conditions of Terms & Privacy, which can be accessed by clicking on the blue link at the bottom of the User Sign up Form.

Click ‘Sign Up’ to complete the account creation.

At any time, users can update their profile by accessing Account Login/Sign-up Interface and logging in using the username and password provided during account creation.

In the case that the user has forgotten their user name and/or password, click ‘Forgot Password’ hyperlink (with grey font that turns red when hovering the cursor over it) in the Account login/sign-up interface (see Figure 1-1). A series of prompts will guide the user through recovery of their user name and/or password. A valid email address linked to the username is required.

When logging into an existing account, the Hydrosimulatics Account Edit Form will appear (see Figure 1-4), which includes the same fields for editing that were included in the Hydrosimulatics Sign Up Form (Figure 1-2), but also additional features related to user accounts. In the bottom-left, information related to the Account Type (Free vs. Premium – see next section), the date the account was created, and the starting and end dates for premium subscriptions¹. Users may also directly view messages sent to their account emails and view recent activity by using the hovering over and clicking the ‘View Message’ or ‘View Activity Report’ hyperlinks just above the Account Type information (seen Figure 1-5).

¹ These will be the same as the ‘Created Date:’ for free users.

Hydrosimulatics Account Edit Form

You can edit/update this form. Click 'View Message' to check your new message.

User Name
NOTE: At least 3 characters and the first character must be a letter, the second and third must be letter or number
 courtaza

Email
 courtaza10@gmail.com

Password (at least more than 3 characters)

Repeat Password

User Type
 Please select one that best describes you:

Affiliation
 Please select one that best describes your affiliation:

GuestPassword (can be shared by multiple users)

Additional Information

First Name: Zachary
 Last Name: Curtis
 Email Address: curtisza10@gmail.com
 Country: US
 Purchase Date: 11/8/2018 10:42:03 AM

Use email notification for comment/message alert

By creating an account you agree to our [Terms & Privacy](#). **Accept**

[View Message](#)
[View Activity Report](#)
[View Available Download List](#)

Account Type: Subscriber1
 Created Date: 08/25/2018 12:09:41 PM
 Start Date: 11/8/2018 10:43:38 AM
 End Date: 12/8/2018 10:43:38 AM
 Subscription Days Left: 30
 Active Account

Figure 1-4: Hydrosimulatics Account Edit Form.

Close Message Box

Payment: ch_3d0f421yxc0mhrchfzsd
 (card brand: Visa with ending number: 4242)
 Purchase Date: 11/8/2018 10:42:03 AM

[View Activity Report](#)
[View Available Download List](#)

Account Type: Subscriber1
 Created Date: 08/25/2018 12:09:41 PM
 Start Date: 11/8/2018 10:43:38 AM
 End Date: 12/8/2018 10:43:38 AM
 Subscription Days Left: 29:99
 Active Account

Figure 1-5: View Message Box

If the 'View Activity Report' option is selected, the user must specify a date (year, month, day) from which to extract an activity report, then click 'Apply'. The report file will be outputted to the text box just below (see Figure 1-6).

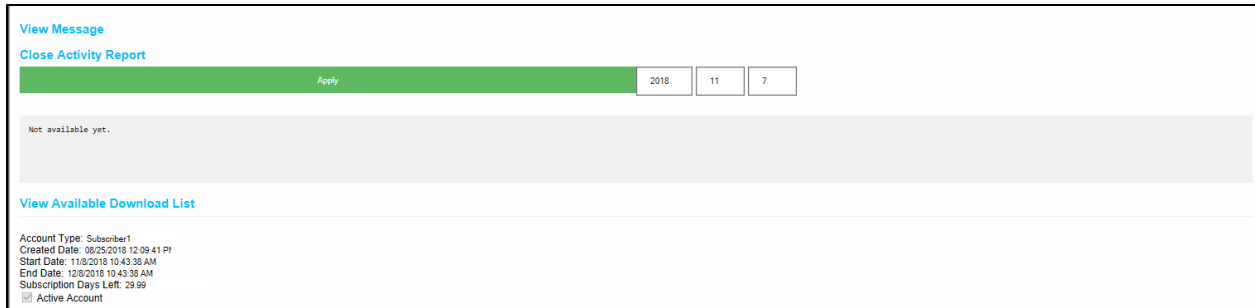


Figure 1-6: Activity Report interface.

If the ‘Click Available Download List’ option is selected, four clickable icons will appear (see Figure 1-7).

- Clicking ‘IGW2D’ allows users to download/run the free-to-use IGW2D.exe application file on the local desktop.
- Clicking ‘Latest Plane View Image (PRJ)’ downloads/opens a *.bmp image file of the latest model results (plan view display of head contours, velocity vectors, particle pathlines, etc.).
- Clicking ‘Latest Model Results Files (txt)’ downloads/opens a MAGNET model result file which includes the model structure (number of cells, cell size, etc.) and model outputs for each cell (cell index, head, conductivity, bottom and top aquifer elevation). See Figure 1-8 for an example.
- Clicking ‘Latest Model Result Files (Binary)’ downloads a *.bin binary file containing model results from the last time-step (or steady-state results), specifically, the 2D or 3D head distribution and solute concentration distributions. This *.bin file can be uploaded to MAGNET to provide initial/boundary conditions for a new model as described Section 3.6 (feature coming soon).

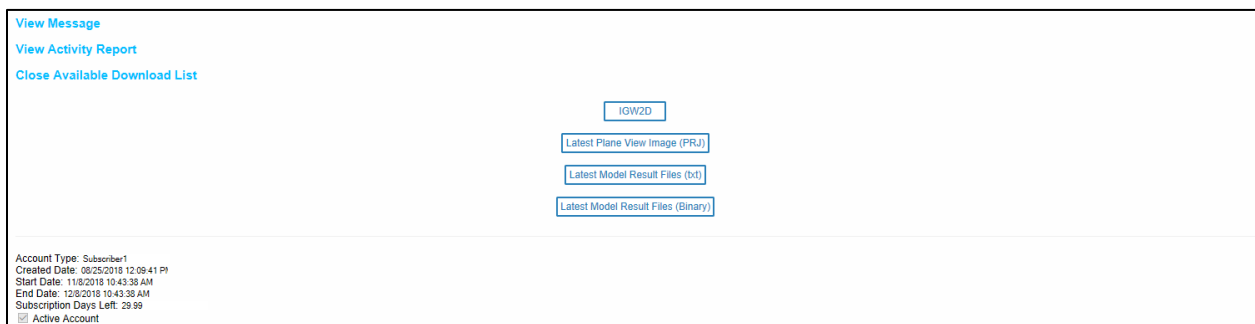


Figure 1-7: MAGNET User Available Download List.

```

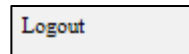
MagnetResult (1) - Notepad
File Edit Format View Help
Magnet Calculated Results for User: curtisza
Model Type=Transient
NX= 40
NY= 36
NZ= 1
DX= 7860.22073440444
DY= 7871.4010290945
Xorg= 448730.460575429
Yorg= 141294.694931794
Tstamp= 44

I, J, K, IB, H, C, TopE, BotE
-----
1,1,1,,113.04,.0000,203.20,112.04
2,1,1,,113.04,.0000,207.20,112.04
3,1,1,,113.04,.0000,214.37,112.04
4,1,1,,113.04,.0000,222.17,112.04
5,1,1,,113.04,.0000,227.88,112.04
6,1,1,,113.04,.0000,235.81,112.04
7,1,1,,113.04,.0000,248.54,112.04
8,1,1,,113.04,.0000,259.11,112.04
9,1,1,,113.04,.0000,264.12,112.04

```

Figure 1-8: Example MAGNET model result file.

Finally, at any time while using MAGNET, the user may log out from their account by selecting the 'Logout' button in the utilities menu.



1.4 Guest Accounts

The Guest Password in the 'My Account' in the submenu can be used to allow others MAGNET users to load and view model results associated with the 'Master' user account, although guests cannot edit/refine the model (e.g., they cannot add conceptual features, change inputs, etc.) This provides an efficient and powerful way for educators/supervisors/project members/etc. to view and analyze models created by students/ interns/other project team members/etc. and provide quick feedback.

The first step is to enter a password in the Guest Password field of the 'My Account' submenu (save changes by clicking the green 'Update' button in the bottom-right corner of the submenu). This is the password to share with others that will review/analyze the model created by the 'Master' account.

To view and analyze an existing model, Guest users should access the login submenu: zoom into a Zoom Level greater than 3, then press 'Simulate'. Then, the 'Master' account username should be entered, and the Guest Password should be entered in the Password field. Make sure to check the box next to 'Guest Account' (see Figure 1-9). Successful login with Guest account credentials will automatically load the latest MAGNET model simulated on the Master account.

For steady-state flow models, the following will be available for the Guest User:

- Plan-view model results (head contours, velocity vectors)
- Model outputs at node location (see section 5.3)
- Cross-section results² and 3D model visualizations (see Sections 5.4 and 5.5)
- Mass Balance and Monitoring Well results (see Sections 6.1-6.2)

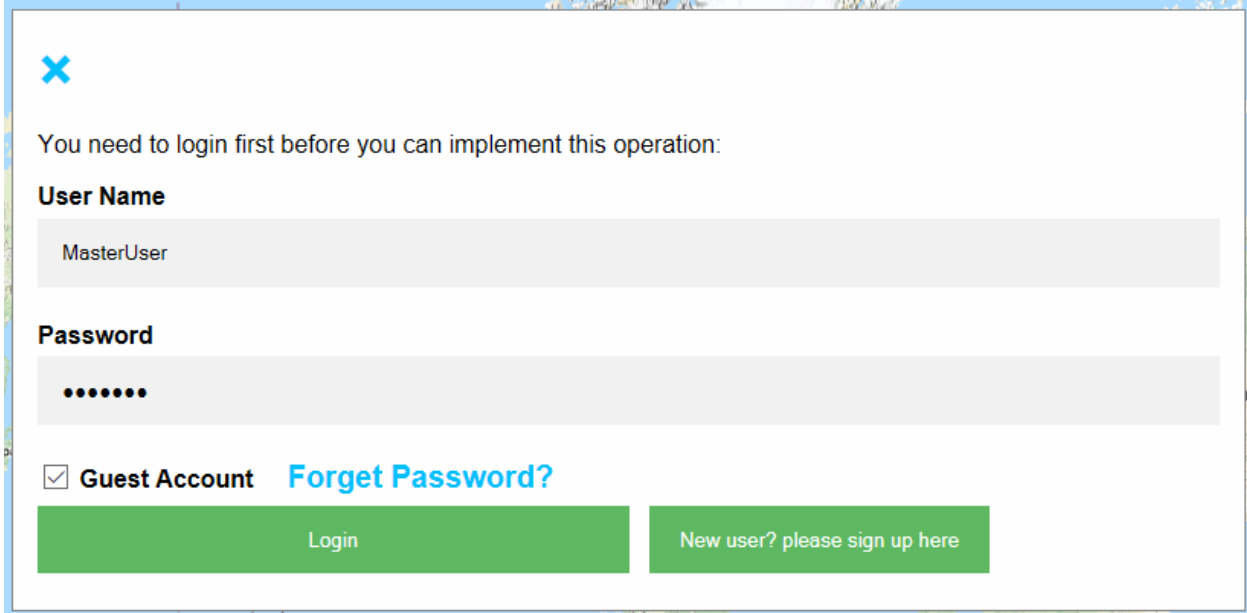


Figure 1-9: Signing in to a 'Guest' account.

For particle tracking and plume transport applications and/or transient flow models, the Guest user may view the results from the last time-step of the master model (i.e., all results available for steady-state flow models) as well as draw new cross-sections³.

If the simulation is still running on the Master account, results observed by Guest user the will be updated in the User MAGNET modeling environment in 'real-time' (almost instantaneously).



1.5 MAGNET Premium

MAGNET Premium is offered for MAGNET projects requiring computationally intensive, realtime, visual modeling capabilities and thier associated cloud computational resources (CPU, memory, and storage). To ensure that the MAGNET online platform can benefit as many people around the world as possible, the advanced, cloud-powered modeling capabilities are made available at the lowest possible costs (similar to the costs required in securing/renting commerical cloud-based computational infrastructure/resources).

² Guest users cannot draw a new cross-section for steady-state models.

³ When the Guest user draws a new cross-section, the results are updated the next time the Master user opens the Display Charts with the 'View Result' button (see section 5.4).

See more details at <https://www.magnet4water.com/overview.html> and <https://www.magnet4water.com/shopping/>.

Chapter 2 **MAGNET MODELING ENVIRONMENT**

The MAGNET Modeling Environment shown in Figure 2-1 is divided into 7 parts (A through G) to facilitate its functionality and explanation of items/buttons/zones/etc. The subsequent sections explain each of these 7 parts of the Modeling Environment.

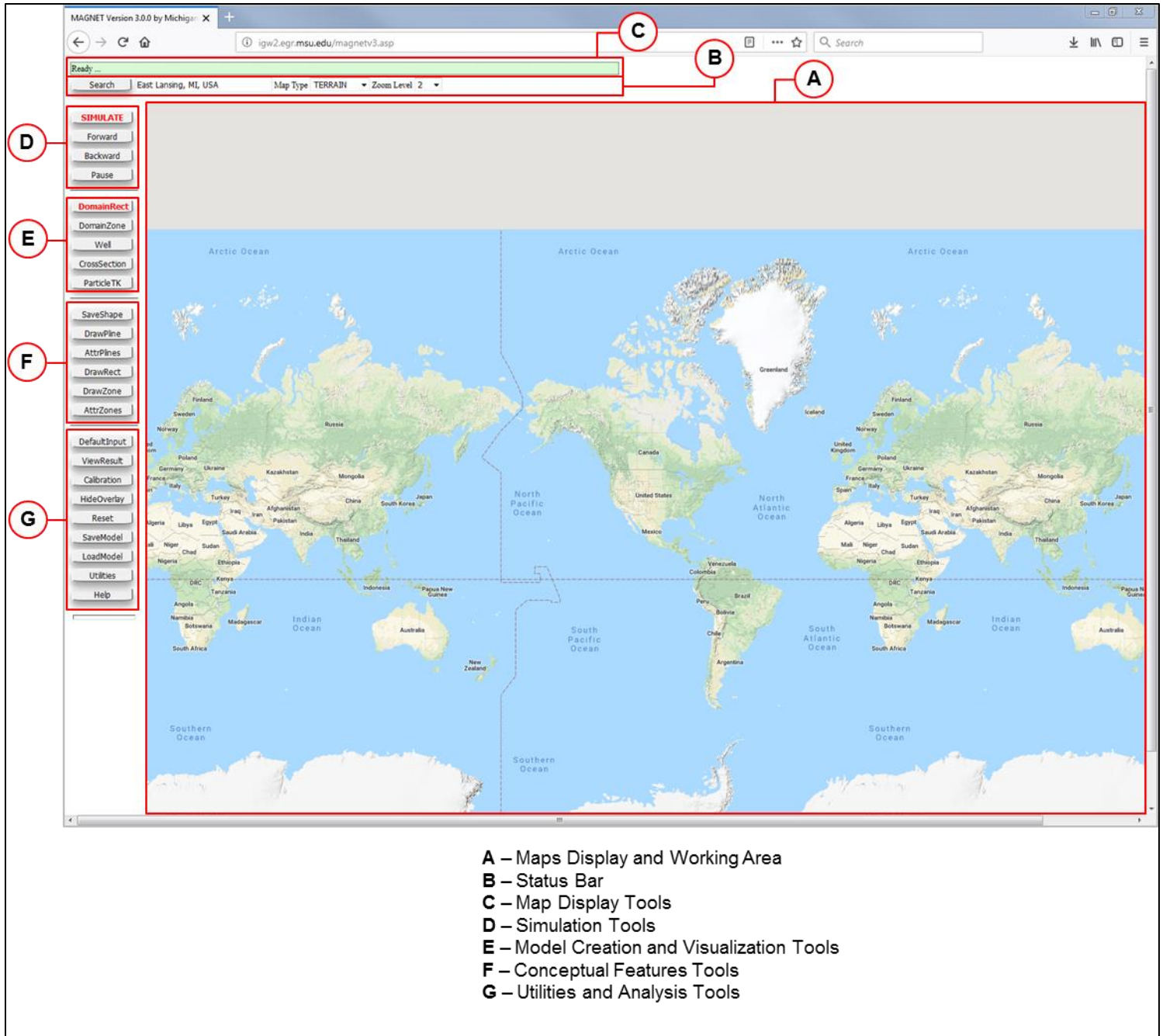


Figure 2-1: MAGNET Modeling Environment upon login.

2.1 Maps Display and Working Area

The Maps Display and Working Area (MDWA) occupies a majority of the MAGNET Modeling Environment. THE MDWA is the region where conceptual modeling is performed and subsequent solutions are obtained (examples are shown throughout this manual). The MDWA is enabled with Google Maps, or alternatively, the user may choose to use Microsoft Bing – although the way to create and edit geometric features (model domain, zones/polygons, polylines) is slightly different. See Section 3.8 for more details.

The default view upon login is a map projection of the entire globe.

The user may navigate to different locations within the MDWA using the Navigation tools (see Section 2.3) or by click-dragging within the display. A double-click on the left mouse (LM) button anywhere will increase the zoom level (i.e., “zoom in”), and a double-click of the right mouse (RM) button decreases the zoom level (zoom out). Display will increase the zoom level (i.e., “zoom in”), and a double-right-click decreases the zoom level (“zoom out”). Additionally, the user can click and hold the Ctrl keyboard button while moving the scroll wheel on the mouse to zoom in or out.



2.2 Status Bar

The status bar is located at the top of the MAGNET Modeling Environment. The status bar is used to display different messages related to the status of modeling jobs submitted to the MAGNET server (see examples in Table 2-1 below).

Table 2-1: Examples of Status Bar messages displayed while modeling with MAGNET.

Ready ...
Connecting to IGW server:#3(1 to 5) and running the model, Please wait ..
Running on IGW server...
Scan images done.. Redrawing Cross-section ...

2.3 Map Display Tools

Just below the Status Bar is a suite of standard map tools to help the user navigate to the area of interest and customize the background image. The user can use the text box to enter the coordinates (i.e., latitude and longitude) or name (formal or approximate) for the area of interest. The spatial extent of the display can be controlled with the ‘Zoom Level’ drop-down menu – increasing the zoom level increase the resolution (and decrease the spatial extent) and vice versa.

The Map Type drop-down menu allows the user to select from four background maps: TERRAIN (default setting); SATELLITE; ROADMAP; and HYBRID (satellite display with roads/names overlaid).

Once changes in the text box and/or the drop-down menus have been made, apply them by clicking on the ‘Search’ button. An example is shown below.

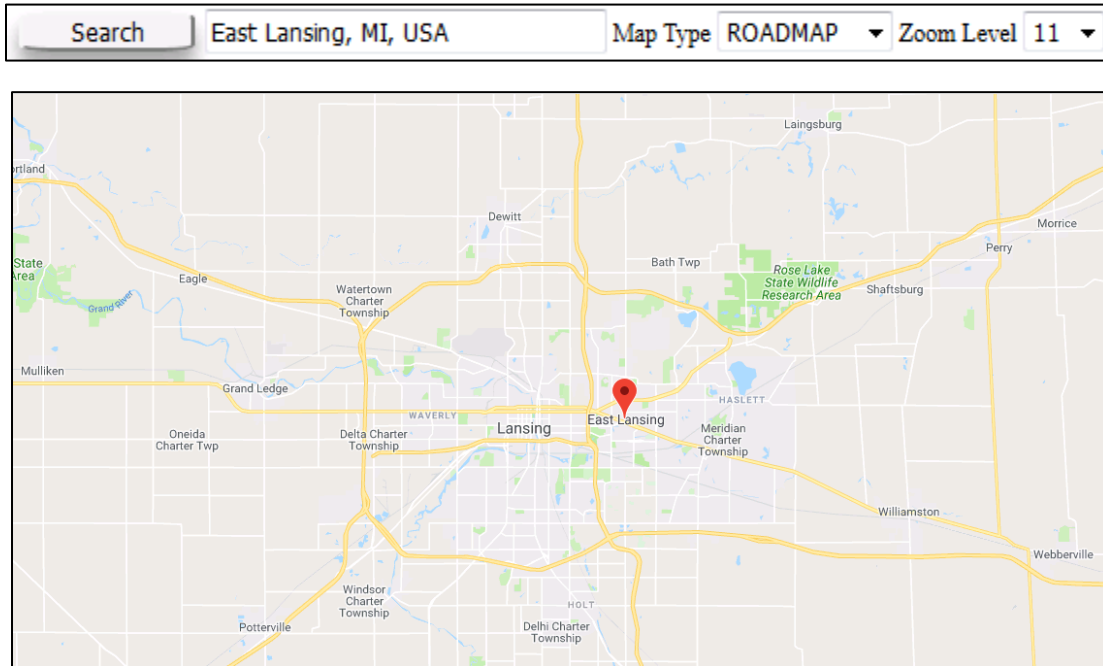
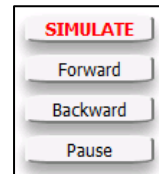


Figure 2-2: Using the Map Display Tools to customize the display.

2.4 Simulation Tools

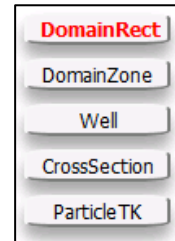
The Simulation Tools are used for executing a conceptual model created in the MDWA. Clicking the red ‘SIMULATE’ button initiates the process of solving the groundwater equations for the domain created by the user. The ‘Forward’ and ‘Backward’ buttons are used to run a transient simulation forward or backward in time, respectively. The ‘Pause’ button pauses a transient simulation after solving the current time-step. The ‘Forward’, ‘Backward’, and ‘Pause’ buttons may also be used during particle tracking applications of a steady-state groundwater solution.



More details related to model simulation are provided in Section 5.1 and elsewhere in this Manual.

2.5 Model Creation and Visualization Tools

The 2nd-tier of buttons along the left-side of the MAGNET Modeling Environment are used for creating and visualizing groundwater models. The red ‘DomainRect’ button allows the user to create a rectangular-shaped modeling domain within the MDWA. Similarly, ‘DomainZone’ allows the user to trace out a polygon of any shape as the model domain. More details are provided in Section 3.1.



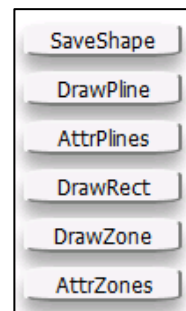
The ‘Well’ button allows users to add a well feature at any location within the model domain. These can be used “monitoring wells” to view and compare model predictions with observations, or they be used to simulate pumping/injection at a point. More details are provided in Section 4.3.

The ‘CrossSection’ button is used to draw a cross-section visualization and analysis along polylines within the model domain. See more details in Section 5.4.

The ‘ParticleTK’ button expands to a sub-menu that includes tools for particle tracking applications useful for visualizing flow paths and groundwater source areas. More details are provided in Section 5.6.

2.6 Conceptual Features Tools

The 3rd-tier of buttons along the left-side of the MAGNET Modeling Environment are used for creating conceptual features such as polylines (e.g., streams, injection trenches, etc.) and zones (e.g., lakes, contaminant sources, polygons for water balance analysis, etc.).



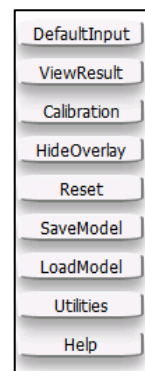
The ‘DrawPline’ button is used for creating a polyline feature within the model domain. The ‘AttrPlines’ is used to edit the attributes of any existing polyline feature. See Section 4.1 for complete details.

The ‘DrawRect’ button creates a rectangular-shaped zone within the model domain. The ‘DrawZone’ button allows users to create zones of any shape. The ‘AttrZones’ is used to edit the attributes of any existing polygon/zone feature. See Section 4.2 for complete details of implementation.

The ‘SaveShape’ button is used to finalize the creation of a polyline or zones created in the model domain – this is a necessary function since the double-click mouse utility is reserved for changing the zoom level in the MDWA.

2.7 Utilities and Analysis Tools

The 3rd-tier of buttons along the left-side of the MAGNET Modeling Environment are used for managing user background images, model files, and model input settings, as well as viewing model results and comparing outputs with observations (model calibration/evaluation).



The ‘DefaultInput’ button provides access to the Default Model Parameters and Options menu (see Figure 2-3). This menu is used to: 1) assign grid and solver properties; 2) parameterize aquifer elevations, hydraulic conductivity and aquifer recharge; 3) apply boundary conditions from a “Parent Model”; 4) define transient simulation settings; and 5) customize particle

tracking and model output display options. This menu will be thoroughly discussed in Chapter 3 and elsewhere in this manual.

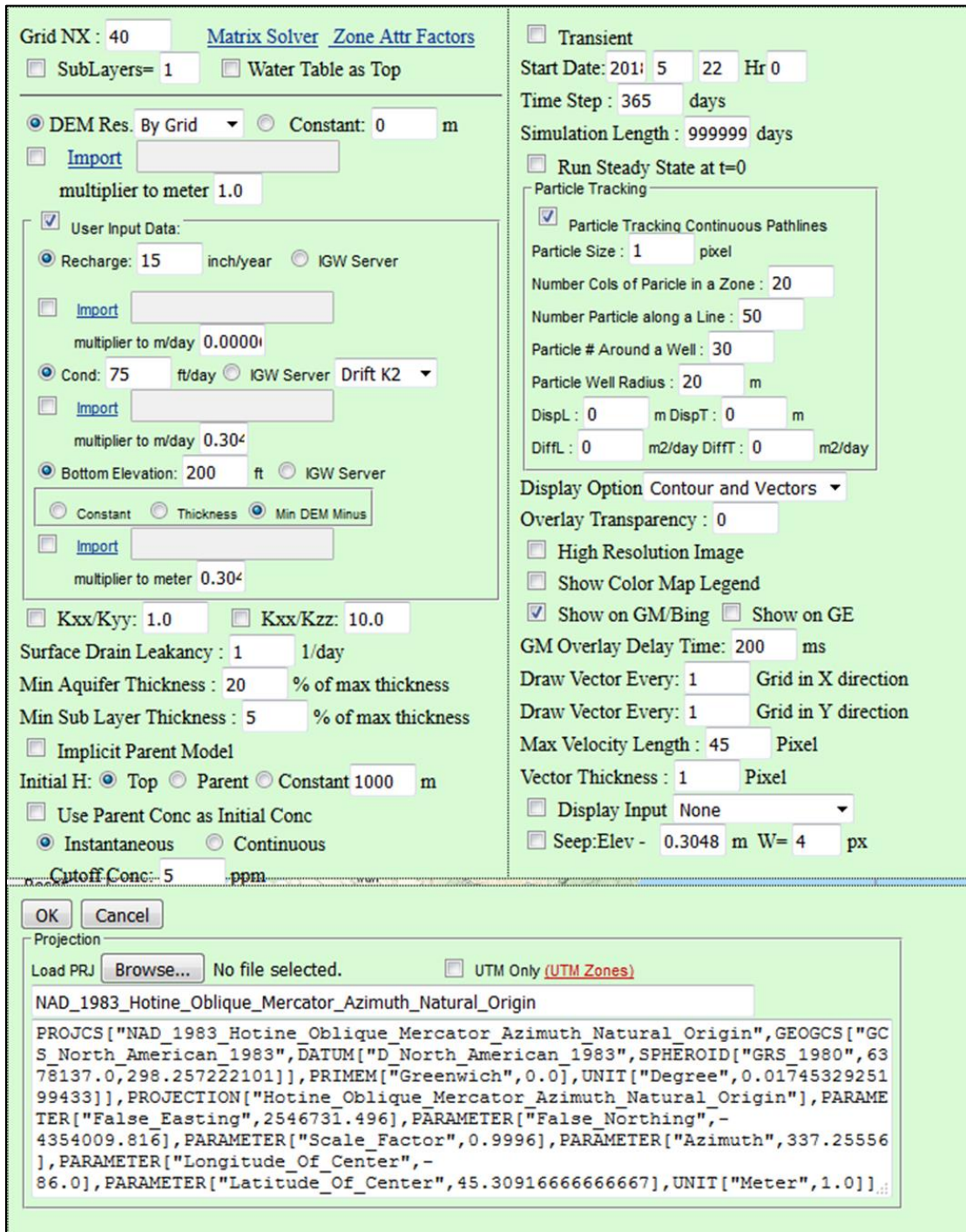


Figure 2-3: Default Parameters and Options menu.

Chapter 3 MODEL CREATION AND PARAMETER OPTIONS

Typically, model creation begins with navigating to the area of interest and creating a domain of sufficient size to capture the regional dynamics (more on this below). Users may opt to generate purely synthetic models (in which none of the global framework are utilized, in which case a user background may be applied (see Chapter 7). This chapter will primarily deal the typical case of using all/some of the spatial framework data during modeling.

3.1 Delineating the Model Domain

After navigating to the area of interest (see Section 2.3), use the red 'DomainRect' button to create a rectangular-shaped modeling domain within the MDWA, or use the 'DomainCreate' button to trace out a polygon of any shape as the model domain (e.g., to follow along a groundwater divide or watershed boundary).



- When drawing a rectangle, indicate the position of the first vertex with a single LM click, and indicate the position of the vertex diagonal from the initial vertex with another single LM click⁴.
- When drawing a polygon, single-click the LM button to define the vertices of the polygon that will define the model domain.

Hover the cursor over the circles at vertex locations and line segment mid-points to enable node editing capabilities. Click-drag a node to adjust the size or shape of the model domain. After any change is made, an 'un-do' button (↶) will appear next to the last node that was edited. Click on ↶ to return to the previous node position. Note changes can only go back one node-edit in the current version of MAGNET.

An example of creating and adjusting the model domain is shown in Figure 3-1.

⁴ Allow for a brief period of time (i.e., at least 2-3 seconds) between LM clicks. Otherwise, the locations of the vertices may not be consistent with what is desired.

Using the **DomainRect** button

Using the **DomainZone** button

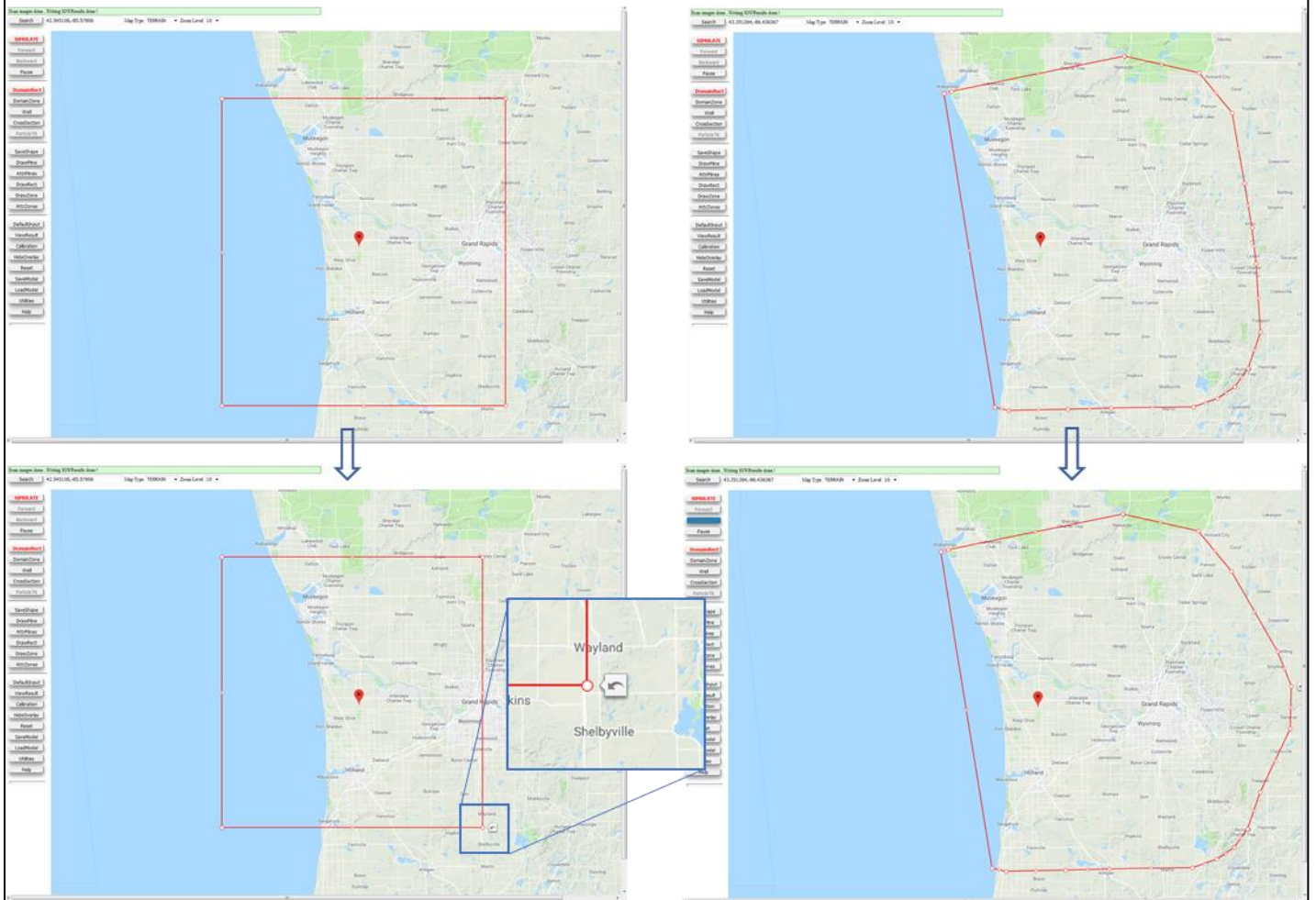


Figure 3-1: Creating and adjusting the model domain using the 'DomainRect' and 'DomainZone' button. The blue in-set map highlights the 'un-do' button available after altering any node position.

Use the 'SaveShape' button to finalize the model domain. This will return the cursor to "navigation mode" (i.e., the cursor assumes the mouse navigation utilities described in Section 2.1). If 'SaveShape' is not utilized, a new rectangle will be started (when using 'DomainRect'), or another vertex will be added to the polygon (when using 'DomainZone').

3.1.1

Note on Domain Size

By default, the conditions along the domain's lateral and bottom boundaries are 'no-flow' (i.e. flux across the boundary is zero at all points). However, lateral boundary conditions can be derived from an existing model encompassing the study area. Thus, an "iterative" procedure is recommended for proper characterization of flow dynamics within the study area (see Section 3.6).

3.2 Model Grid and Solver Settings

In the top-left portion of the Default Parameters and Options menu, the user can select/input a number of model grid options. This includes: the number of model nodes in the horizontal (west-east) direction (NX), the number of sub-layers in the vertical direction (i.e., into the earth), matrix solver settings, and multiplicative factors for spatially-explicit raster files of hydraulic conductivity and recharge (see Figure 3-2).

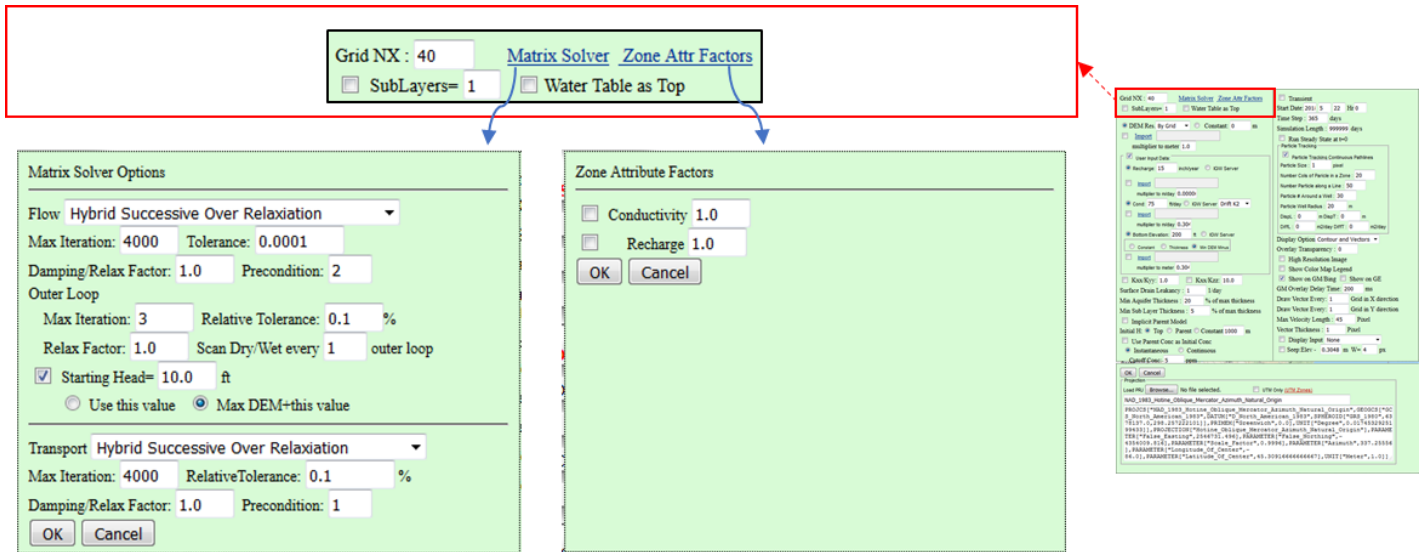


Figure 3-2: The Model Grid and Solver settings available in the Default Parameters and Options menu. Default values/settings are shown.

The number of nodes in the north-south direction is automatically computed based on NX and the shape of the model domain.

MAGNET allows for vertical discretization of geological (conceptual) layers into desired number of computational layers. Subdividing can be done based on 1) the land surface as the top boundary surface (this is the default setting); or 2) with a water table solved by a previously simulation within the study area (check the box next to ‘SubLayers=’ to use this setting). In the latter case, the previous water table used for sub-dividing must cover the entire model domain, but need not be the same size. However, what is most common is to:

- first simulate the model domain without sub-dividing the aquifer in the vertical direction;
- then check the box next to ‘Water Table as Top’ in the Default Parameters and Options menu, save the changes (click ‘OK’), and re-run the simulation.

The latest water table can also be saved by clicking the ‘SaveModel’ button along the left side of the MAGNET Modeling Environment, then selecting ‘Latest Water Table’ from the submenu. To load this water table, check the box next to ‘Import’, then click Import to browse to and select the saved water table file.

In the Basic version of MAGNET, the maximum value of NX is limited, and the number of geological (conceptual layers) is restricted to one, and the number of computational layers is restricted to three. No

such restrictions apply for the Premium version of MAGNET. The Premium version does not limit grid size nor the number of sub-layers in the vertical direction.

3.2.1 *Zone Attributes Factors*

Click the ‘Zone Attr Factors’ link to open the Zone Attribute Factors sub-menu. This menu is used to apply multiplicative factors for spatially-explicit rasters of hydraulic conductivity and recharge. For example, if a factor of 1.5 is used, every value in every raster cell is multiplied by 1.5. The user must check the box(es) next to the parameter (Conductivity or Recharge) and then specify the multiplicative factor in the text-box to the right. Click ‘OK’ to save changes.

3.2.2 *Matrix Solver Settings*

Click the ‘Matrix Solver’ link to open the Matrix Solver Options sub-menu. This menu includes parameters and options for the numerical scheme(s) and algorithms used to solve mathematical groundwater problem formulated for the study area. The algorithms available for solving the discretized flow and/or transport equations include:

- Hybrid Successive Over Relaxation
- Successive Over Relaxation
- Conjugate Gradient
- Bi-Conjugate Gradient
- Bi-Conjugate Gradient with Partial Pivoting

The user may specify the convergence criterion for the numerical solutions, namely: the maximum number of iterations; the threshold of acceptable relative percent error (between successive iterations); the damping/relaxation factor; and the number of preconditions (or condition number) The user may also customize the ‘outer loop’ matrix solve scheme by specifying the number of maximum iterations, the threshold of acceptable relative percent error, the Relaxation Factor, and the frequency with which the dry/wet cells are scanned in the solution matrix.

The reader should consult a reference on matrix solver algorithms described above (see [sparse matrix solvers](#))

The default grid and matrix solver settings are shown in Figure 3-2.

3.3 *Aquifer Elevations*

The top and bottom elevations of the model can be assigned in a number of different ways using the Default Parameters and Options menu. Figure 3-3 shows the options/parameters that are available, which are discussed in the following subsections.

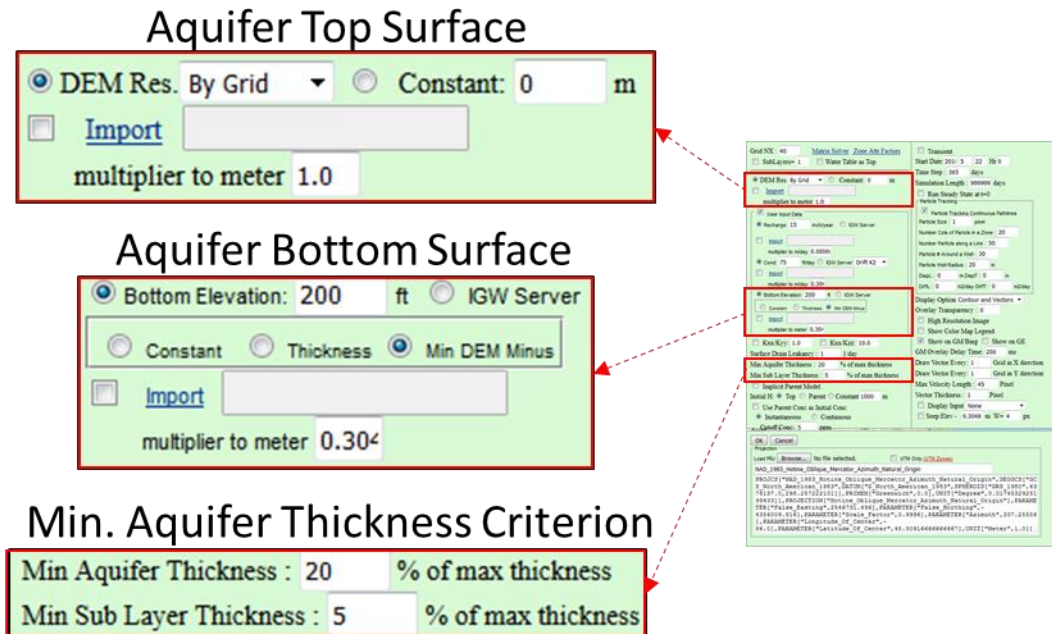


Figure 3-3: Options and parameters for assigning aquifer elevations in MAGNET. Default values/settings are shown.

3.3.1 Top Elevations

By default, the top elevation of the model follows the DEM of a resolution consistent with the model cell size (e.g., if the cell size is $\geq 90\text{m}$, the 90m resolution will be used, if the cell size is $\approx 30\text{ m}$, the 30m resolution will be used etc.). The user may also choose from 90m, 30m, 10m, and 1m (for select regions) resolutions by accessing the drop-down menu next to ‘DEM Res.’

Choose the ‘Constant’ option and use the text box to enter the elevation to be used as the top boundary throughout the model domain.

User raster files can also be used to represent the aquifer’s top surface. To do this, check the box next to the ‘Import’ link, then select the link to open a file browser. Browse to and select the appropriate file for importing. A prompt with file metadata will appear after the file has been uploaded, and the file path and name will appear in the text box next to ‘Import’. If the units of the DEM are not meters, use the text-box next to ‘multiplier to meter’ to enter the appropriate multiplier, e.g., to convert from ft. to m, enter 0.3048 , as $1\text{ ft.}=0.3048\text{m}$.

Caution!. Note that, in most areas, 1m DEM is currently not available. If the default option is used for assigning DEM as the top elevation (‘By Grid’), and the domain size is sufficiently small such that the grid size is on the order of 1 m, MAGNET will produce ‘bad’ (unphysical) results. this problem can be alleviated by utilizing 10 m DEM rather than the default DEM setting⁵.

⁵ Alternatively, users may input their own DEM file as a raster input (see above).

The bottom boundary of the model domain (aquifer bottom) can be assigned as:

- a constant value prescribed by the user (see Figure 3-4a);
- a surface following the aquifer top elevation surface such that the aquifer has a constant thickness (Figure 3-4b);
- a constant elevation that is a prescribed value below the minimum DEM elevation within the model domain (Figure 3-4c) (this is the default setting);
- a surface following the bedrock top surface raster on the IGW Server (Figure 3-4d); and
- a surface based on a user bedrock top surface raster file, which can be uploaded by checking the box next to 'Import' and then selecting the link to open a file browser. Browse to and select the appropriate file for importing (Figure 3-4e).
- a surface prescribed by a spatially-variable aquifer thickness from the IGW Server (Figure 3-4e)
- a surface prescribed by a spatially-variable aquifer thickness from a user aquifer thickness raster file (Figure 3-4f).

The value in the text box will be applied for the first three options in the appropriate manner, e.g., if 200 ft is in the text box, and the 'Thickness' option is selected, 200 ft will be used as the value to be subtracted from top elevation nodal values across the model domain.

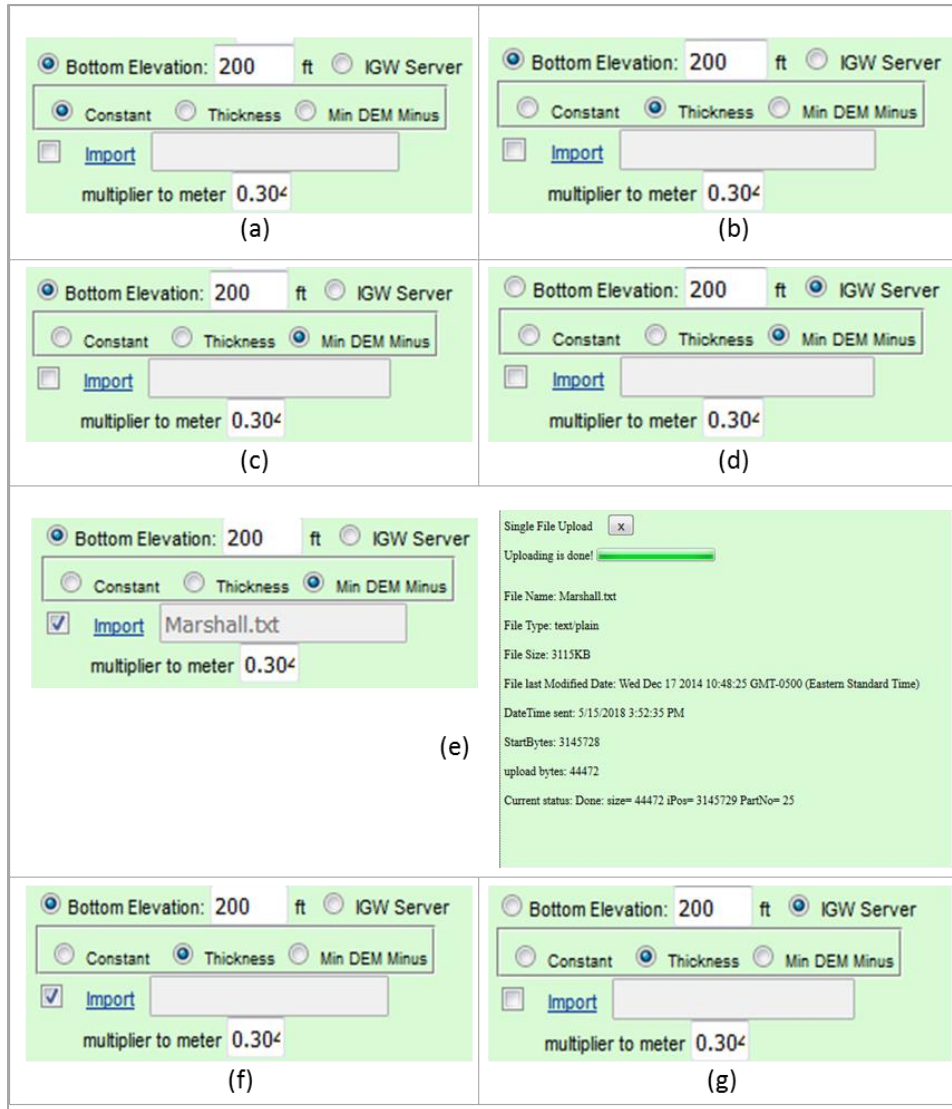


Figure 3-4: Different possible settings for the aquifer bottom elevations.

3.3.2 *Note on Minimum Aquifer Thickness*

When there is significant variability in thickness of the aquifer layer (or computational layers in the case of vertical sub-dividing), issues may arise related to numerical instability of the solution process. To alleviate/avoid such issues, a minimum aquifer (or computational) layer thickness may be assigned (see Figure 3-3) as a percentage of the maximum aquifer thickness. For example, if the default value of 20% is applied for the minimum aquifer thickness, any areas of the model domain with a thickness less than 20% of the maximum thickness will adopt a thickness equal to the value representing 20% of the maximum thickness.

Of course, the value assigned by the user which is largest enough to avoid numerical issues (but small enough so that key details are not lost) depends on the range of aquifer thickness variability and the numerical scheme used when solving the groundwater problem. The default values shown in Figure 3-3 are recommended (especially for beginners), as they yield relatively robust solutions regardless of location.

3.4 Hydraulic Conductivity

A deterministic approach is used for parameterization of hydraulic conductivity (K) of the aquifer. Users assign/input horizontal (K_{xx}) values and implicitly assign vertical (K_{zz}) and north-south (K_{yy}) conductivities through prescribing vertical and horizontal anisotropy ratios (K_{xx}/K_{zz} and K_{xx}/K_{yy} , respectively).

Users can assign a constant effective K_{xx} value for the entire model domain, or they can use the spatially-explicit raster from the IGW Server (see Figure 3-5).

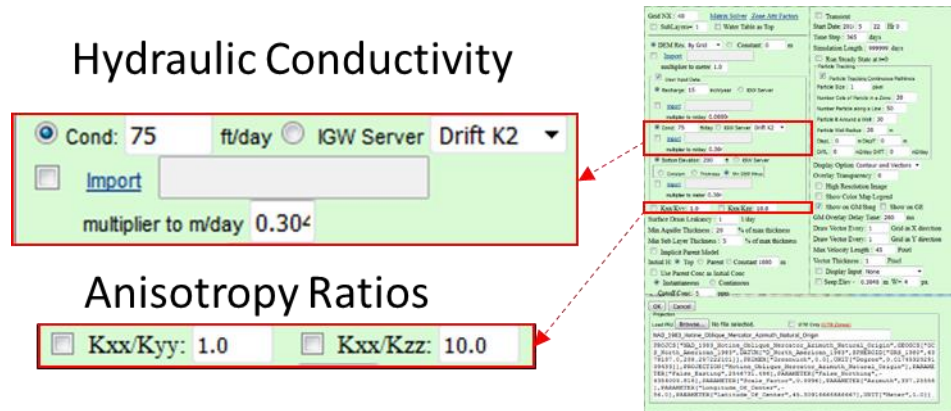


Figure 3-5: Options and settings for parametrizing the hydraulic conductivity of the aquifer. Default values/setting are shown.

User raster files can also be used to assign K_{xx} values across the model domain. To do this, check the box next to the 'Import' link, then select the link to open a file browser. Browse to and select the appropriate file for importing. A prompt with file metadata will appear after the file has been uploaded, and the file path and name will appear in the text box next to 'Import'.

Note that zones may be added in the model domain to assign a value of K_{xx} that overrides the value(s) provided in the Default Parameters and Options menu (see Section 4.2.1)

3.4.1 Anisotropy Ratios

By default, the aquifer system is assumed to be isotropic (i.e., $K_{xx}/K_{zz} = K_{xx}/K_{yy} = 1$). To apply vertical and/or anisotropy ratios, check the box(es) next 'Kxx/Kyy' and/or 'Kxx/Kzz' and enter the desired ratio(s) in the text box(es) (see Figure 3-5).

3.5 Recharge and Surface Seepage

In MAGNET, water may be added to the aquifer through the top boundary as recharge (precipitation/surface water that infiltrates down to the water table) or can be removed through the top boundary as surface seepage (in the case where the hydraulic head is larger than the land surface elevation).

Recharge to the aquifer is treated as specified flux [$L T^{-1}$] that is constant in time⁶. Users can specify a single value to be applied through the model domain (default setting), or can use the spatially-variable recharge raster available from the IGW server ((currently available in the State of Michigan, U.S.A.; global raster K coming soon).

User raster files can also be used to assign recharge values across the model domain. To do this, check the box next to the ‘Import’ link, then select the link to open a file browser. Browse to and select the appropriate file for importing. A prompt with file metadata will appear after the file has been uploaded, and the file path and name will appear in the text box next to ‘Import’. If the units of the recharge raster are not meters per day, use the text-box next to ‘multiplier to m/day’ to enter the appropriate multiplier, e.g., to convert from in./yr to m/day, enter 0.0000696, as 1 in./yr. = 6.96×10^{-5} m/day.

Note that zones may be added to the model to assign a recharge value that overrides the value(s) provided in the Default Parameters and Options menu (see Section 4.2)

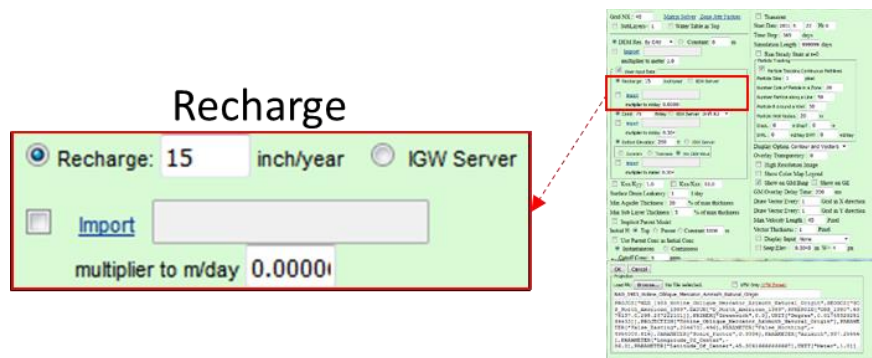


Figure 3-6: Options for applying recharge along the top of the model boundary. Default values/settings are shown.

Surface seepage is computed as the amount of hydraulic head exceeding the land surface (computed as aquifer head minus land elevation) multiplied by the leakancy (a factor representing hydraulic conductivity per unit thickness of the land surface). The default value is 1 day^{-1} .

To remove surface seepage as a possible sink of groundwater for topmost model cells, set the leakancy to zero.



Figure 3-7: Surface leakance parameter text box.

⁶ Interactive Groundwater (IGW) includes a module for process-based and non-process-based watershed modeling, which enables time-dependent recharge simulation.

In Section 3.1.1, it was mentioned that, by default, the conditions along the domain’s lateral and bottom boundaries are ‘no-flow’, and that lateral boundary conditions can be derived from an existing model encompassing the study area.

The general idea is to: first create (see Section 3.1) and simulate (Section 5.1) a steady-state model that contains – and is a few times bigger than – the area of interest. This is referred to as the “Parent” model. Then, create a smaller, nested model (or series of models) that derives its/their boundary conditions from the parent model(s). The box next to ‘Implicit Parent Model’ must be checked in the Default Input Parameters and Options menu after drawing the “Child” model domain within the Parent Model domain. Interpolation of Parent model head values will be applied along the boundary of the (finer-grid) Child model.

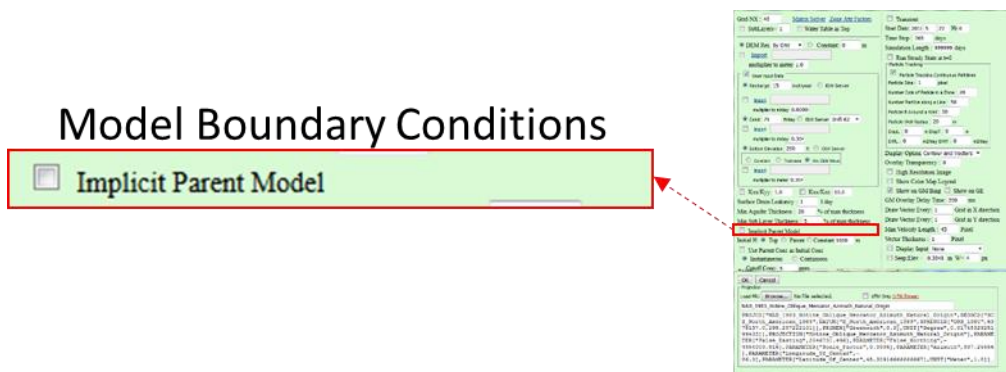


Figure 3-8: Options for applying boundary conditions for flow and/or solute transport modeling.

3.6.1

Hierarchical Modeling Example

An example of the hierarchal modeling approach for applying boundary conditions is shown in Figure 3-9. The initial model (top-left of Figure 3-9) is chosen to be much larger than the area of interest so that the major regional “groundwater mounds” can be identified and used to guide submodel delineation. The idea is to incrementally zoom in to the area of interest – at each step using the location of groundwater mounds to place groundwater boundaries, in the assumption that flow patterns further beyond the groundwater mounds have little impact on the area of interest. Note that once ‘DomainRect’ or ‘DomainZone’ is selected for delineating the submodel boundaries, the red polygon of the parent model domain is removed from the MDWA. Once the new (sub) model is submitted for simulation, the results of the parent model are removed from the model domain.

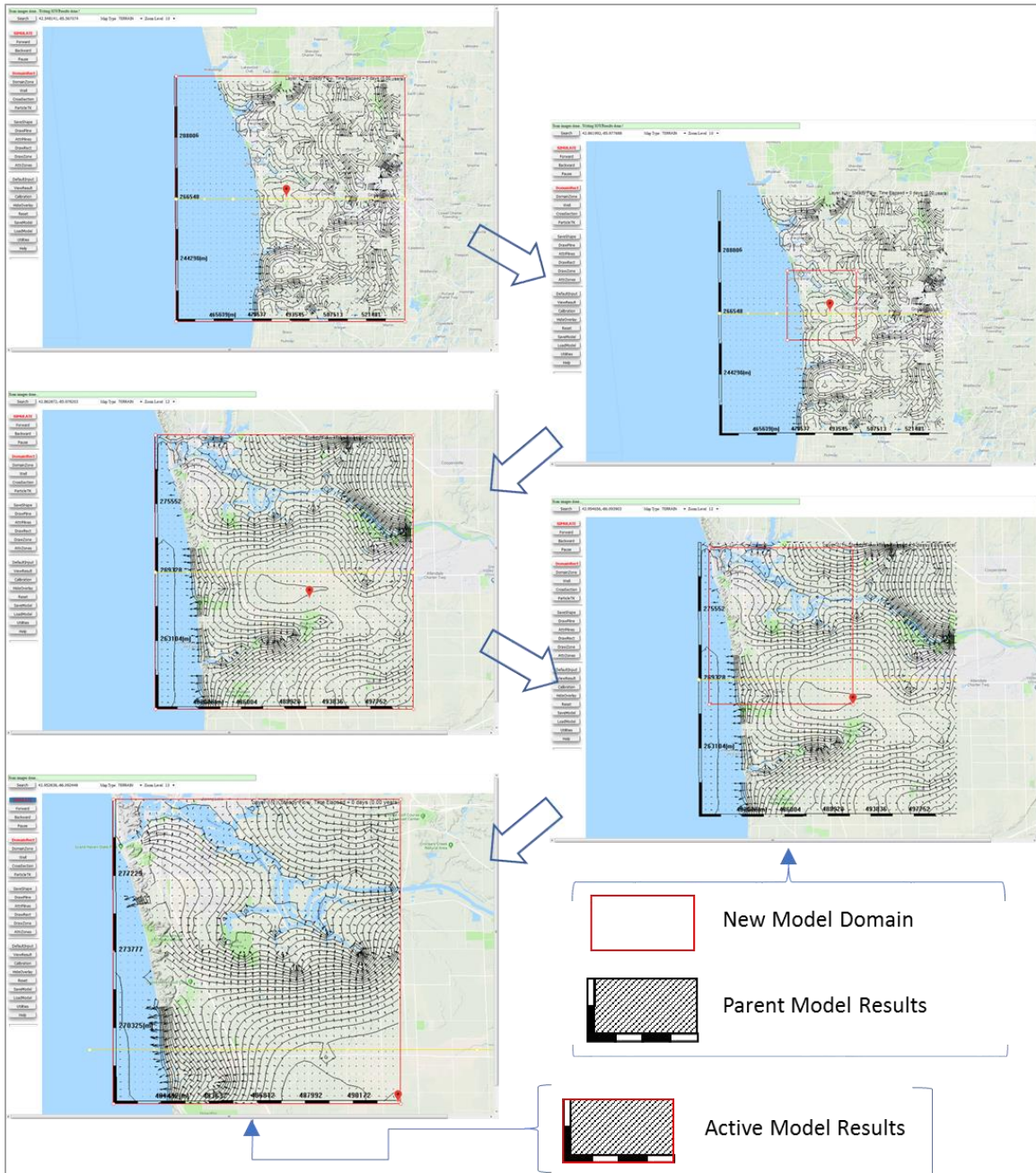


Figure 3-9: Example of the hierarchical modeling approach for developing appropriate boundary conditions in the study area.

3.7 Transient Model Settings & Initial Conditions

By default, flow simulations in MAGNET are run at steady state. To perform unsteady (transient) modeling in MAGNET, check the box next to 'Transient' in the upper-right part of the Default Parameters and Options Menu (see Figure 3-10). Required inputs are the start date and time (hour of the day), the time step,

and the simulation length. Use the text boxes next to ‘State Date’ (year, month, day), ‘Hr’ (hours), ‘Time Step:’ (days), and ‘Simulation Length’ (days) to define user inputs.

Check the box next to ‘Run Steady State at t=0’ to use the head distribution solved for steady state conditions as the initial head distribution of the transient simulation (see Figure 3-10).

Other options for the initial head distribution are shown in Figure 3-11. The ‘Top’ option will use the Top Elevation surface assigned by the user (see Section 3.3.1). The ‘Parent’ option uses the head distribution of a larger Parent model spanning the new (Child) model domain (see Section 3.6). Head values are interpolated from the (coarse-grid) Parent model to the (finer-grid) Child model. The ‘Constant’ option will assign the value in the text box as a constant head value across the entire domain.

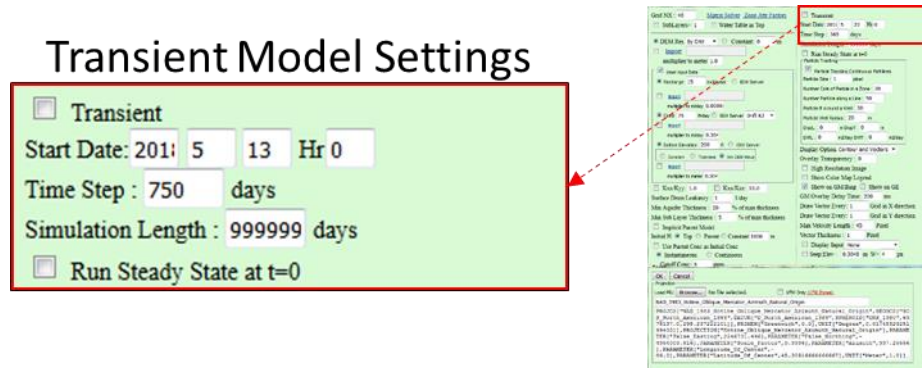


Figure 3-10: Options for unsteady (transient) modeling in MAGNET. Default values/parameters are shown.

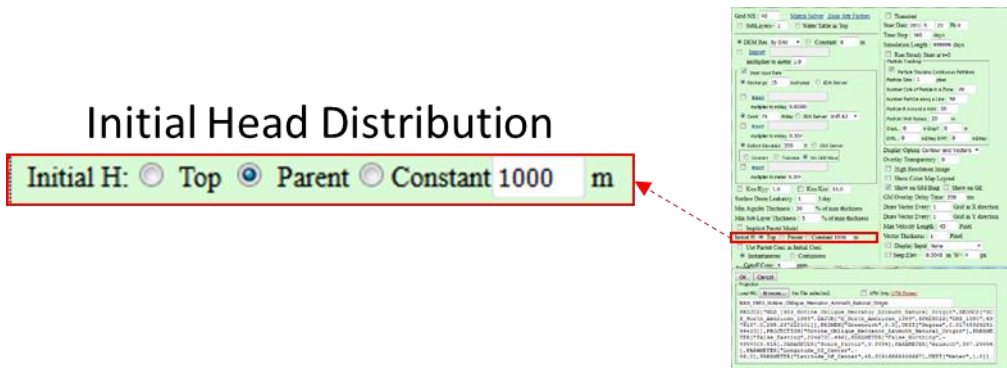


Figure 3-11: Options for assigning an initial head distribution across the model domain. Default value/parameters are shown.

Chapter 4 CONCEPTUAL FEATURES: LINES, ZONES & WELLS

After creating the model domain and finalizing the model inputs/parameters/options (or simply applying the default settings), additional conceptual features may be added within the model domain. This includes: polylines (e.g., streams, injection trenches, etc.), zones (e.g., lakes, contaminant sources, polygons for water balance analysis, etc.), and extraction/injection wells. Each of the features is discussed in detail in the following subsections.

A4.1 Line Features & Attributes

To add a polyline feature to the model domain, LM click on the ‘DrawPline’ button along the left-side of the MAGNET Modeling Environment (see Section 2.1). The cursor will become a cross-hair, allowing for placement of polyline vertices with single-clicks of the LM button⁷. The polyline will appear as a series of green line segments with circles at the vertex locations (see Figure 4-1). The location can of any vertex can be changed by hovering the mouse over it and click-dragging to the new desired location⁸.

Once the last vertex has been placed, LM click the ‘SaveShape’ button to finalize the polyline. This will launch the Polyline Attributes menu (see Figure 4-1). Several options are available for the polyline conceptual feature: prescribed head boundary condition (constant, variable, transient, or equal to the water table elevation); head-dependent flux boundary condition (one-way or two-way) or as a prescribed flux (constant or transient).

If the polyline needs to be discarded, LM click the ‘Delete’ button to close the Polyline Attributes menu and remove the polyline feature from the MDWA.

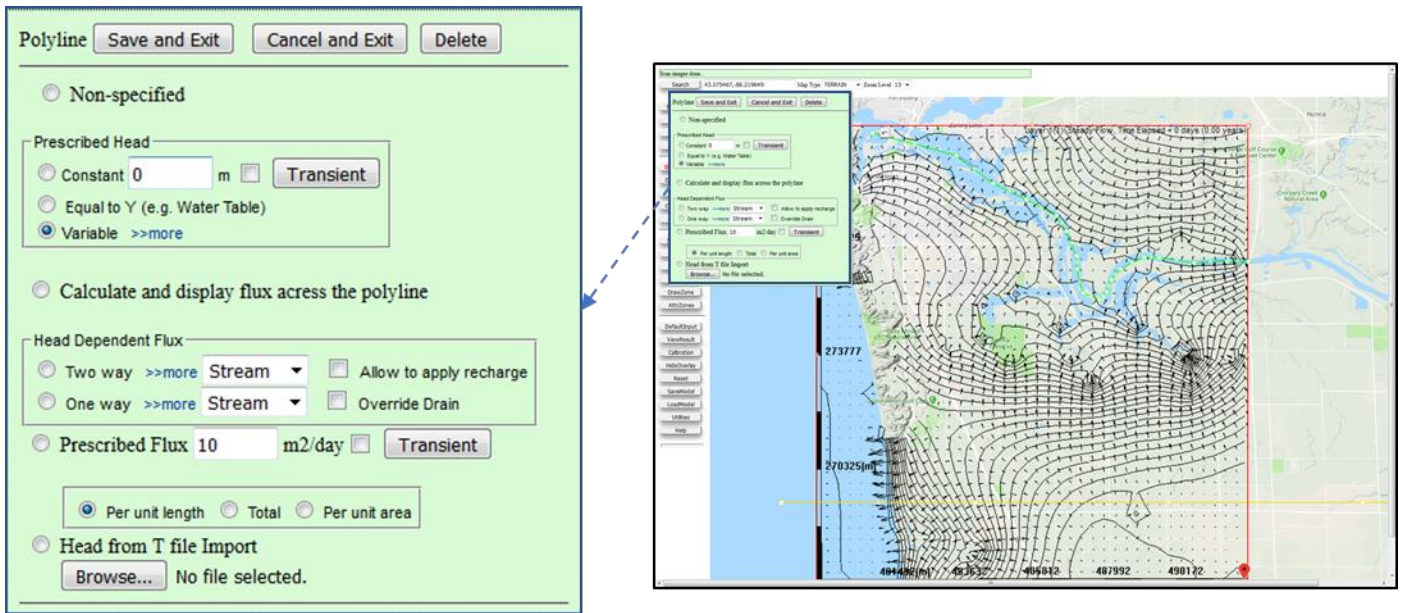


Figure 4-1: Polyline feature added to the model domain and the Polyline Attributes menu.

4.1.1 Prescribed Head Options

Different prescribed head options can be applied using the Prescribed Head option box. The simplest treatment is a constant values at all vertex locations (and at all time-steps, if running a transient simulation).

Spatially-constant prescribed head. Select the radio button next to ‘Constant’, and enter the desired head value (in meters) in the text box. To assign spatially-constant but time-dependent head values, leave the ‘Constant’ radio button selected, but also check the radio box next to ‘Transient’. Then LM click the ‘Transient’ button to open the sub-menu for editing transient head data (see Figure 4-2). Indicate the starting

⁷ Allow for a brief period of time (i.e., at least 2-3 seconds) between LM clicks. Otherwise, the locations of the vertices may not be consistent with what is desired.

⁸ Any vertex location can also be updated after finalizing the polyline.

date and time with the text boxes next to 'State Date' (year, month, day), 'Hr' (hours). Then, provide the following inputs for each timestep: TimeN – or the number of days since the starting date and time; and ValueN – the head value corresponding to TimeN. The head values may be entered in any units; however, if meters are not used, make sure to enter the appropriate multiplier in the text box next to 'Multiplier to meter:'. For example, to convert from ft. to m, enter 0.3048, as 1 ft.=0.3048m.

After all desired changes have been made, LM click the 'OK' button. Or click 'Cancel' to discard all changes and close the sub-menu.

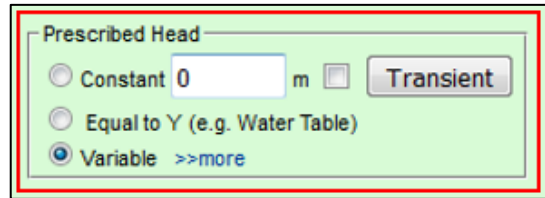
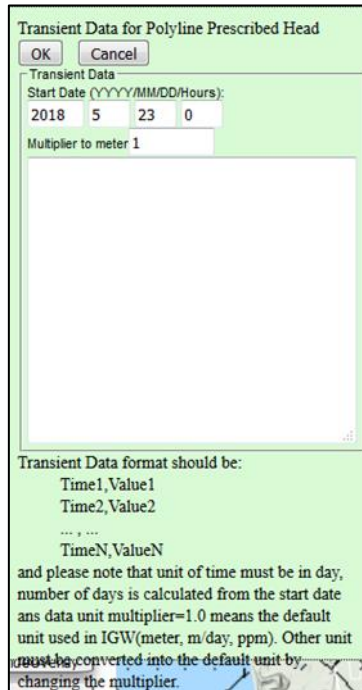


Figure 4-2: Sub-menu for editing Transient prescribed head data.

Spatially-variable prescribed head. Multiple options are available if a spatially-variable prescribed head is desired. First, one may select the radio button next to 'Equal to Y (e.g., Water Table) to assign a head value equal to the water table elevation at each point along the polyline.

Another option is to select the radio button next to 'Variable' option from the Prescribed-Head option box, then LM click the '>>more' link to open the Edit Polyline Attributes sub-menu (see Figure 4-3). This sub-menu provides information on the Vertex ID (VertexID) and the longitude and latitude coordinates (Long(degree) and Lat(degree), respectively).

By default, the head value at each location is: the top aquifer elevation, as indicated by the checked box next to 'Head = TopE minus' and the value of 0 in the corresponding text-box shown in Figure 4-3. An offset may be applied by entering a non-zero number in the text box. If the head is known at each vertex location, the user can edit the head value (in meters) by LM clicking anywhere within a cell in the Head(m) column (note that each row corresponds to a different VertexID). After all cells have been updated, uncheck the box next to 'Head = TopE minus'.

After all desired changes have been made, LM click the 'Save and Exit' button. Or click 'Cancel and Exit' to discard all changes.

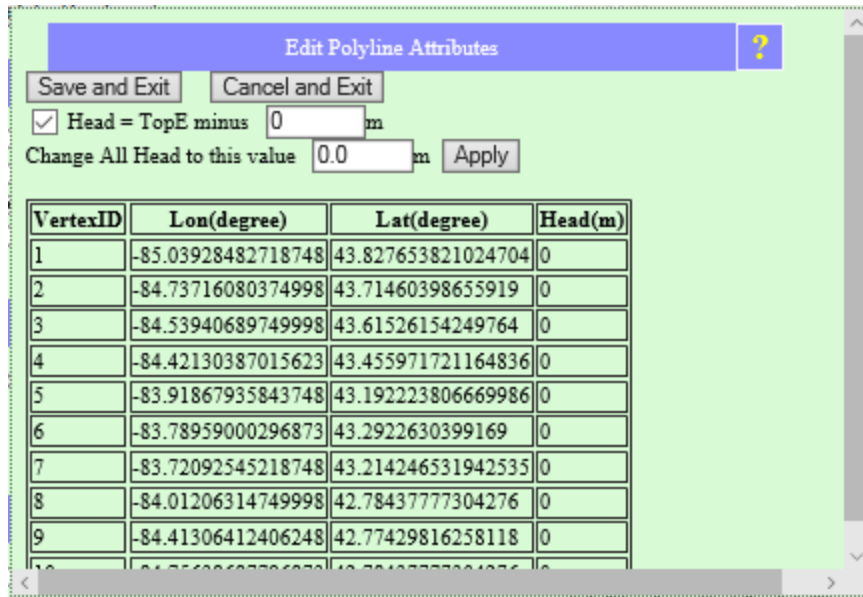
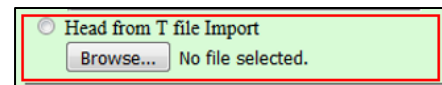


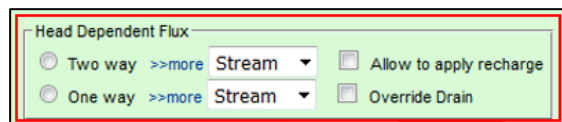
Figure 4-3: Edit Polyline Attributes sub-menu for assigning unique head values at different vertex locations. Default values/options are shown.

Transient head from a T File. This feature is not available in the current version of MAGNET.



4.1.2 Head-Dependent Flux Options

Two-way or one-way head-dependent flux boundary conditions may be applied for polyline features. One-way head dependent flux allows water to leave the aquifer system in the case that the hydraulic head exceeds the top aquifer elevation, but water may not enter the aquifer system (e.g., surface seepage to a drain or ditch). “Two-way” allows for water to enter or leave the aquifer system depending on the direction of the head gradient between the surface water stage and the aquifer head.



Two-way head-dependent flux. Select the radio button next to ‘Two way’ in the Head Dependent option box to apply two-way head-dependent flux boundary conditions. LM click the ‘>>more’ link to open the Edit Polyline Attributes sub-menu (see Figure 4-4). By default, the stage along the polyline is equal to the aquifer top elevation, as indicated by the checked box next to ‘Stage = TopE minus’ and the value of 0 in the corresponding text-box shown in Figure 4-4. An offset may be applied by entering a non-zero number in the text box.

If the stage is known at each vertex location, the user can edit the stage value (in meters) by LM clicking anywhere within a cell in the Stage(m) column⁹ (note that each row corresponds to a different VertexID). Similarly, the leakancy (Leak(m/day)), bottom elevation (BotE(m)), and stream order (Order) can be

⁹ Be sure uncheck the box next to ‘Stage = TopE minus’ if specifying stage at each vertex location.

customized for any cell in the table. Use the drop-down menu to select a column in which all values will be set to the value entered in the corresponding text box. Click ‘Apply’ to update the selected column.

After all desired changes have been made, LM click the ‘Save and Exit’ button. Or click ‘Cancel and Exit’ to discard all changes.

VertexID	Lon(degree)	Lat(degree)	Stage (m)	Leak (m/day)	BotE (m)	Order
1	-71.12379001743943	42.48844440025625	0	2	-1	6
2	-71.12422989971787	42.488919093817664	0	2	-1	6
3	-71.12478779919297	42.489433341110725	0	2	-1	6
4	-71.12507747776658	42.48969441888762	0	2	-1	6
5	-71.1252598679796	42.48987638154224	0	2	-1	6
6	-71.12559246189744	42.490074166436	0	2	-1	6
7	-71.12613963253648	42.4903273301874	0	2	-1	6
8	-71.1266117013231	42.490501379672125	0	2	-1	6
9	-71.12697648174912	42.490596315550555	0	2	-1	6

Figure 4-4: Edit Polyline Attributes sub-menu for assigning polyline attributes needed for computing two-way head dependent flux. Default values/options are shown.

One-way head dependent flux. The steps to apply One-way head dependent flux are similar to that of two-way. Select the radio button next to ‘One way’ in the Head Dependent option box, then LM click the ‘>>more’ link to open the Edit Polyline Attributes sub-menu (see Figure 4-5). By default, the “drain elevation” (i.e., the elevation used to compute the head gradient) is equal to the aquifer top elevation, as indicated by the checked box next to ‘Stage = TopE minus’ and the value of 0 in the corresponding text-box shown in Figure 4-5. An offset may be applied by entering a non-zero number in the text box.

If the drain elevation is known at each vertex location, the user can edit the drain elevation values (in meters) by LM clicking anywhere within a cell in the BotE(m) column (note that each row corresponds to a different VertexID). Similarly, the leakancy (Leak(m/day)) and stream order (Order) can be customized for any cell in the table. After all cells have been updated, uncheck the box next to ‘Drain = TopE minus’. After all desired changes have been made, LM click the ‘Save and Exit’ button. Or click ‘Cancel and Exit’ to discard all changes.

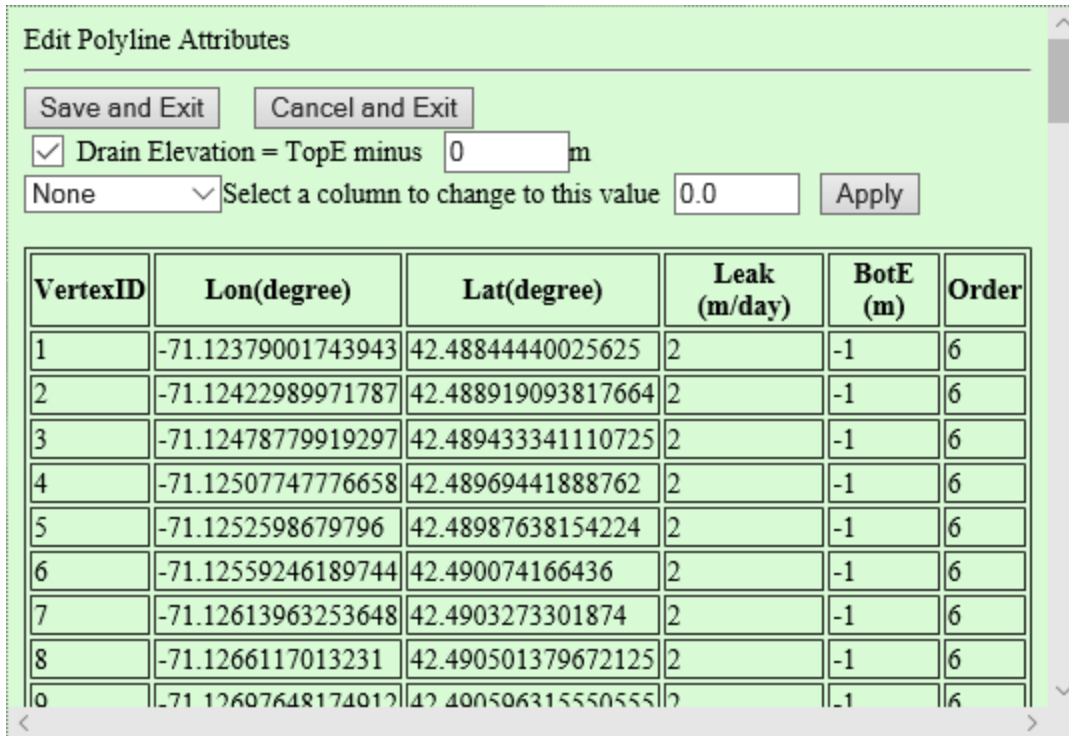
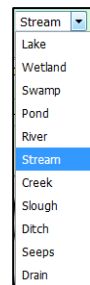


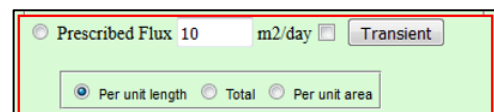
Figure 4-5: Edit Polyline Attributes sub-menu for assigning polyline attributes needed for computing one-way head dependent flux. Default values/options are shown.

The drop-down menus next to each '>>more' link provide different naming option for the polyline feature that will be used in displaying results, e.g., when performing water balance analyses (see Section 6.2). By checking the radio box next to 'Allow to apply recharge', the cell-specific recharge will be applied to cells in which the polyline feature occupies (e.g., when using the one-way head-dependent flux option). By checking the radio box next to 'Override Drain', this allows the polyline (used as a boundary condition) to override surface seepage (only one sink is allowed per cell). If this radio box is left unchecked, surface seepage will apply and the polyline boundary conditions will be ignored.



4.1.3 Prescribed Flux Options

Users may assign a prescribed flux boundary condition along the polyline by selecting the 'Prescribed Flux' radio button and choosing one of the units of input: 1) 'Per unit length' – prescribed flux per unit length (square meters per day) along a polyline (e.g., in the case where stream thickness is negligible), which is the default setting; 2) 'Total' – total prescribed flux across the entire polyline-aquifer interface; and 3) 'Per unit area' – the prescribed flux per unit area along the polyline. Note that all three options apply for steady conditions (i.e., no changes in flux with time) and spatially-constant flux along the polyline.



To assign time-dependent (total) prescribed flux boundary conditions, leave the 'Prescribed Flux' radio button selected, but also check the radio box next to 'Transient'. Then **LM** click the 'Transient' button to open the sub-menu for editing transient flux data (see Figure 4-6). Indicate the starting data and time with the text boxes next to 'State Date' (year, month, day), 'Hr' (hours). Then, provide the following inputs for

each timestep: TimeN – or the number of days since the starting date and time; and ValueN – the flux value corresponding to TimeN. The flux values may be entered in any units; however, if cubic meters per day is not used, make sure to enter the appropriate multiplier in the text box next to ‘Multiplier to m³/day’. For example, to convert from ft³/day. to m³/day, enter 0.0283, as 1 ft³ = 0.0283 m³.

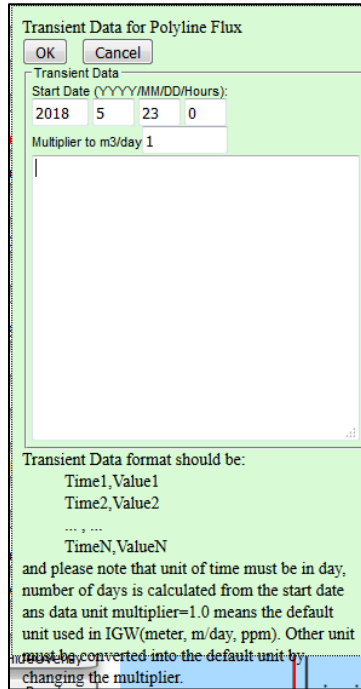
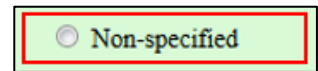


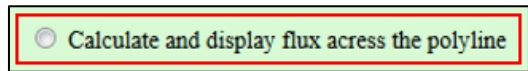
Figure 4-6: Sub-menu for editing Transient prescribed head data.

4.1.4 *Display and Analysis Options*

Polylines need not be used for the purposes of assigning boundary conditions. To use them simply for display purposes (e.g., for highlighting features in the background map or simulation results), select the radio button next to ‘Non-specified’ at the top of Polyline Attributes menu. This is useful for highlighting features in the background map or simulation results.



‘Calculate and display flux across the polyline’: this feature is not available in the current version of MAGNET (coming soon).



4.1.5 *Editing Attributes of Existing Polylines*

When the simulation is complete/paused, any pre-existing polyline feature can be edited by clicking the ‘AttrPlines’ button from left-side of the MAGENT Modeling Environment. This turns all pre-existing polylines red within the model domain (see Figure 4-7). Click once on the polyline to be edited, and select ‘OK’ from the prompt to confirm editing. This opens the Polyline Attributes menu for updating parameters/options of the polyline feature. Click ‘Save and Exit’ to apply the changes to the polyline feature, and continue editing other polyline features, if desired. Once all changes have been



made, click on the ‘AcceptEdits’ button on the left-side of the MAGNET Modeling Environment. This applies all updates and turns the polylines back to green objects in the MDWA.

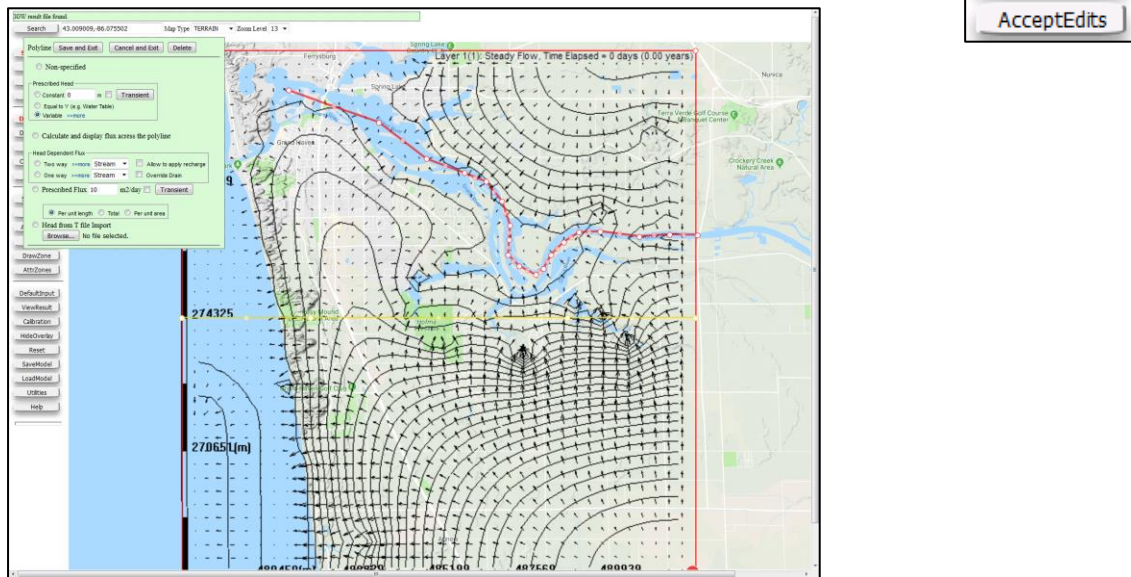


Figure 4-7: Editing attributes of a pre-existing polyline feature.

4.2 Zone Features & Attributes

Zones can be added to the model domain to:

- Perform zone budget (water balance) analyses;
- Assign flow, elevation, and recharge properties/values different from those assigned in the Default Input and Parameters menu (see Figure 2-3) for a subregion of the model domain;
- Define diffusion, dispersion, retardation and decay parameters/coefficients for the purposes of solute transport modeling;
- Create two-way head dependent sources/sinks (e.g., a lake), one-way head dependent sinks (e.g., a groundwater spring or a groundwater-dominated wetland), and prescribed head sources/sinks; and
- Create solute sources defined by a prescribed mass flux or source concentration.

MAGNET will assign zone attributes to every cell in which the center node lies inside of the defined zone area. If two zones overlap, the attributes of smaller zones will take precedence over larger zones. Any cells outside of zones added to the model domain will utilize the attributes/parameters assigned in the Default Parameters and Options menu (see Chapter 3).

To add a zone feature to the model domain, **LM** click on the ‘DrawRect’ button to create a rectangular-shaped zone within the MDWA, or use the ‘DrawZone’ button to trace out a polygon of any shape as the zone feature (e.g., to follow along a wetland or lake boundary).

- When drawing a rectangle, indicate the position of the first vertex with a single LM click in the MDWA, and indicate the position of the vertex diagonal from the initial vertex with another single LM click¹⁰.
- When drawing a polygon, single-click the LM button to define the vertices of the polygon that will define the zone extent.

The zone will appear in with circles at the vertex locations (see Figure 4-8). The location can of any vertex can be changed by hovering the mouse over it and click-dragging to the new desired location¹¹

Hover the cursor over the circles at vertex locations and line segment mid-points to enable node editing capabilities. Click-drag a node to adjust the size or shape of the model domain. After any change is made, an ‘un-do’ button (↶) will appear next to the last node that was edited. Click on ↶ to return to the previous node position. Note changes can only go back one node-edit in the current version of MAGNET.

Once the last vertex has been placed, LM click the ‘SaveShape’ button to finalize the zone. This will launch the Zone Attributes menu (see Figure 4-8). This menu consists of 6 sub-menus: ‘Flow Property’; ‘Elevation’; ‘Transport Property’; ‘Biochemical’; ‘Sources and Sinks Head Dependent’; and ‘Sources and Sinks Prescribed’. The following subsection provide more details on each sub-menu

If the zone needs to be discarded, LM click the ‘Delete’ tab to close the Zone Attributes menu and remove the zone feature from the MDWA.

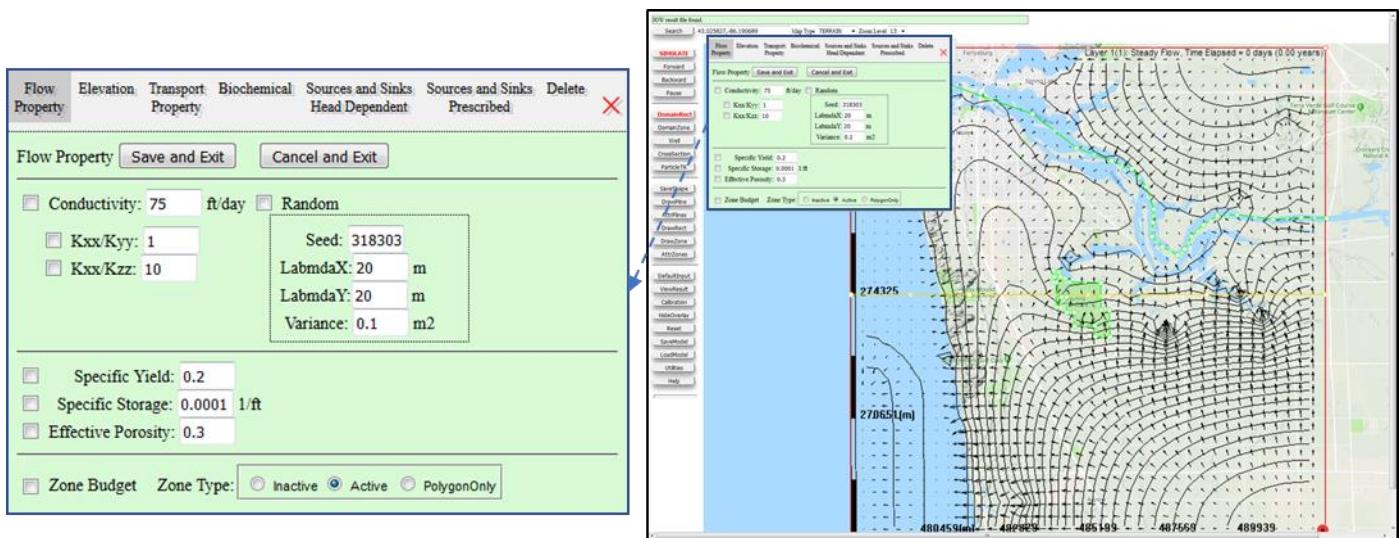


Figure 4-8: Zone feature added to the model domain and the Zone Attributes menu.

4.2.1 Flow Property Attributes

¹⁰ Allow for a brief period of time (i.e., at least 2-3 seconds) between LM clicks. Otherwise, the locations of the vertices may not be consistent with what is desired.

¹¹ Any vertex location can also be updated after finalizing the zone.

Click on the 'Flow Property' tab title to access the Flow Property sub-menu (see Figure 4-8). This sub-menu may be used to assign:

- hydraulic conductivity (K_{xx}) values and anisotropy ratios (K_{xx}/K_{yy} and K_{xx}/K_{zz}) different from those assigned for the rest of the model domain, or even generate a random K_{xx} field defined by characteristic scales of variance;
- specific yield and specific storage values needed for transient modeling; and
- effective porosity values needed for particle tracking and solute transport modeling.

Check the radio boxes next to 'Conductivity' and 'Kxx/Kyy' and/or 'Kxx/Kzz' to use the values entered the relevant text boxes to use zone-specific K_{xx} values and/or K_{xx}/K_{yy} and/or K_{xx}/K_{zz} values.

To generate a random K field, check the radio box next to 'Random'. The 'Seed:' field allows the user to change the random field by entering different numbers. 'LambdaX' and 'LambdaY' are horizontal and vertical correlation scales, respectively, which are a measure of the spatial distances (x- and y-directions) across which the aquifer properties are supposed to be correlated. 'Variance' is a measure of how variable the aquifer properties are with respect to the mean. A higher variance means higher variability of aquifer properties and vice versa.

Check the radio boxes next to 'Specific Yield' to apply the value (dimensionless) assigned in the corresponding text box.

Check the radio boxes next to 'Specific Storage' to apply the value (ft^{-1}) assigned in the corresponding text box.

The Flow Property Attributes sub-menu can also be used to 1) create a water budget of the zone added to the model domain, 2) assign part of the model domain as inactive; or 3) to make the polygon 'virtual' (i.e., all of the applied will be ignored during simulation).

Check the radio box next to 'Zone Budget'. When the model is simulated/run forward, a zone budget for the zone(s) will be available in the Mass Balance Bar chart when analyzing model results with the 'ViewResult' button (see Section 6.2).

Select the radio button next to 'Inactive' to set assign the portion of the model domain covered by the zone as inactive.

Select the radio button next to 'PolygonOnly' to make the polygon 'virtual' (i.e., all of the applied will be ignored during simulation).

4.2.2 *Elevation Attributes*

Click on the 'Elevation' tab title to access up the Elevation sub-menu (see Figure 4-9). Check the radio box next to 'Top Elevation' to use a constant value as the elevation aquifer top surface for the cells over which the zone extends. Check the radio box next to 'Bot Elevation' to use the value in the text box to use a bottom elevation within the zone that is different from the bottom elevation assigned using the Default

Parameters and Options menu (see Section 3.3). The bottom boundary of the model within the zone can be assigned as:

- a constant value prescribed by the user (choose the radio button next to ‘Constant’);
- a surface following the aquifer top elevation surface such that the aquifer has a constant thickness (choose the radio button next to ‘Thickness’); or
- a constant elevation that is a prescribed value below the minimum DEM elevation within the model domain (choose the radio button next to ‘Min DEM Minus’).

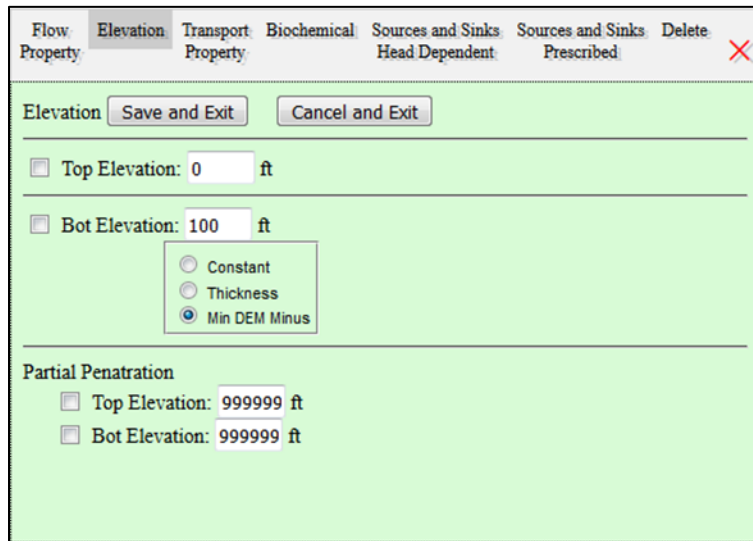
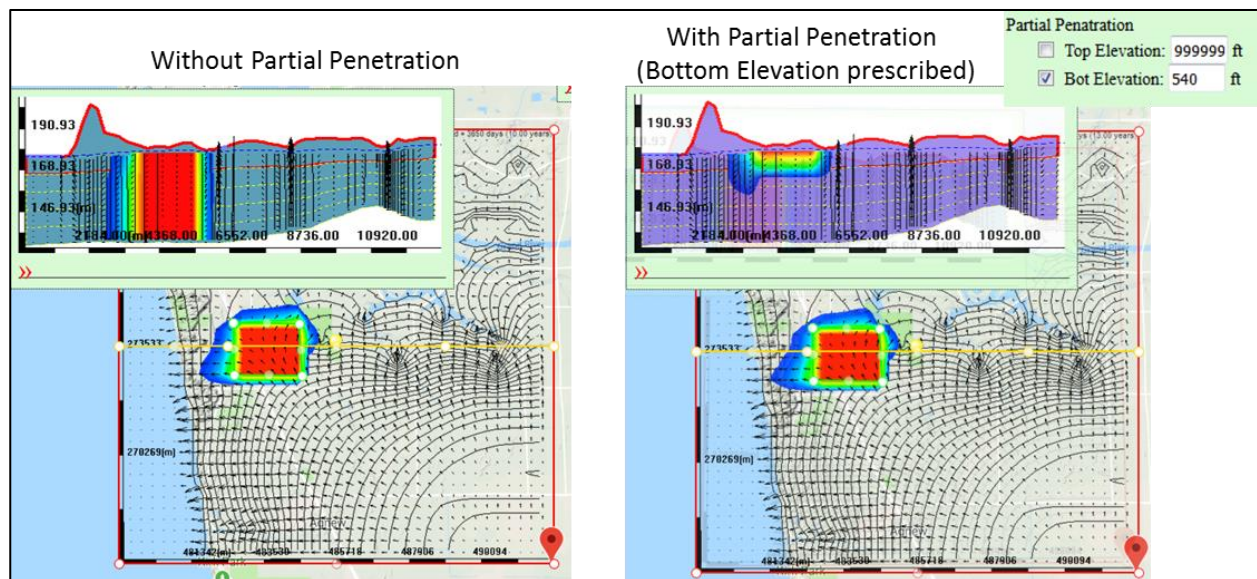


Figure 4-9: Elevation sub-menu (for Zone conceptual features).

The ‘Partial Penetration’ options for the Top Elevation and Bottom Elevation (‘Bot Elevation’) allow zone features to partially penetrate a prescribed thickness of the aquifer¹². For example, users may wish to restrict a contaminant source to the topmost portion of the aquifer system (see Figure 4-10), or insert a “clay lens” of lower hydraulic conductivity at some locations in the subsurface.

¹² Partial penetration only applies when multiple (vertical) sub-layers are used (see Section 3.2).

Check the radio box(es) next to ‘Top Elevation’ and/or ‘Bot Elevation:’ to apply partial penetration elevation extents for the top and bottom elevation of the zone feature, respectively. The default values of ‘99999’ mean that the elevation will be extracted from the domain-wide elevation inputs. Enter different



values in the text boxes provided to use values different than the domain-wide inputs. If one of the radio boxes is left unchecked, the domain-wide inputs will be used (see Figure 4-10).

Figure 4-10: Example Cross-section results (see Section 5.4) of using and not using Partial Penetration for a zone feature in MAGNET. When partial penetration is not used, the solute source is distributed throughout the aquifer system (i.e., across all computational layers). When it is used, the solute source is restricted to the topmost computational layer.

4.2.3 *Transport Property Attributes*

Click on the ‘Transport Property’ tab title to access the Transport Property sub-menu (see Figure 4-9). This sub-menu allows for assigning molecular diffusion and local dispersion coefficients potentially useful for solute transport modeling. Check the radio box next to ‘Molecular Diffusion L’ and/or ‘Molecular Diffusion T’ and input non-zero molecular diffusion coefficients (units: m^2/day) to include lateral (L) and transverse (T) molecular diffusion in the solute transport simulations. Similarly check the radio box next to ‘Local Dispersion L’ and/or ‘Local Dispersion T’ and input non-zero local dispersion coefficients (units: m^2/day) to include lateral (L) and transverse (T) local dispersion in the solute transport simulations.

Solute source concentrations and extents can be added using zones and the Sources and Sinks tabs in the Zone Attributes menu (see subsections 4.2.5 and 4.2.6).

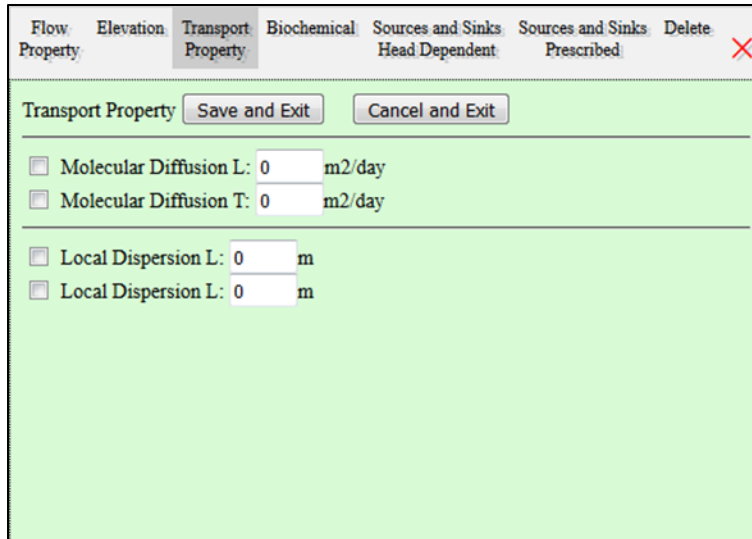


Figure 4-11: Transport Property sub-menu (for Zone conceptual features).

4.2.4 Biochemical Attributes

Click on the 'Biochemical' tab title to access the Biochemical Property sub-menu (see Figure 4-9). This sub-menu allows for assigning retardation and 1st-order decay coefficients potentially useful for solute transport modeling (see Figure 4-12). Check the radio box next to 'Retardation' to simulate the retardation of a solute due to biochemical processes or soil:water partitioning. By default, a retardation factor of 1 is used during simulation. Update the value in the text box next to 'Retardation Factor', or select the radio box next to 'Partitioning Kd:' to use a partitioning coefficient to simulate partitioning of a solute to the solid (sorbed) phase during transport. If a soil density (g/m³) is known, check the radio box next to 'Soil Particle Density:' and enter the known value in the provided text box.

MAGNET allows for simulating reactive decay of a pollutant (1st-order) by checking the radio box next to 'First Decay:'. The rate of reaction can be controlled by assigning a first order decay coefficient (units of days⁻¹) or a known half-life (days). For the former, select the radio button next to 'Coefficient ' and enter a non-zero value in the provided text box. For the latter, select 'Half Life' and enter a non-zero value in the provided text box.

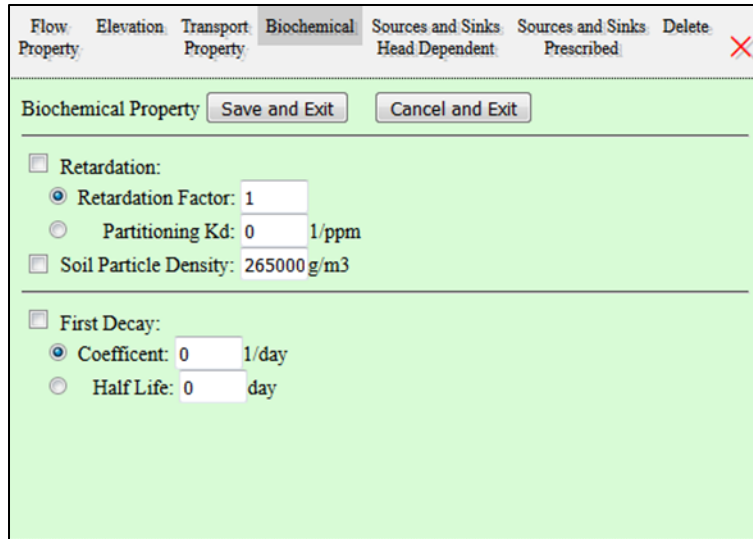


Figure 4-12: Biochemical sub-menu (for Zone conceptual features).

4.2.5 Sources and Sinks (Head Dependent)

Click on the ‘Sources and Sinks Head Dependent’ tab title to access the Head Dependent Flux sub-menu (see Figure 4-13). This sub-menu is used to assign the zone as a two-way or one-way source or sink of water, or as a “general head-dependent flux” (see more below in this subsection).

Two-way head-dependent flux. To assign the zone as a two-way source (e.g., a lake or pond), check the radio box next to ‘Two-way’. The drop-down menu next to ‘Name’ provides different naming options for the zone feature that will be used in displaying results, e.g., when performing water balance analyses (see Section 6.2). The stage of the water source/sink can be assigned as:

- a constant value that does not change with time – select the radio button next to ‘Const:’ and enter the desired value)
- a constant value that is time-varying – after selecting the radio button next to ‘Const:’, check the radio box next to the ‘Transient’ button
- the aquifer top elevation minus some prescribed value; by default, the stage is set to the aquifer top elevation, as indicated by the value of zero in the text box next to ‘Top E – ’ shown in Figure 4-13.

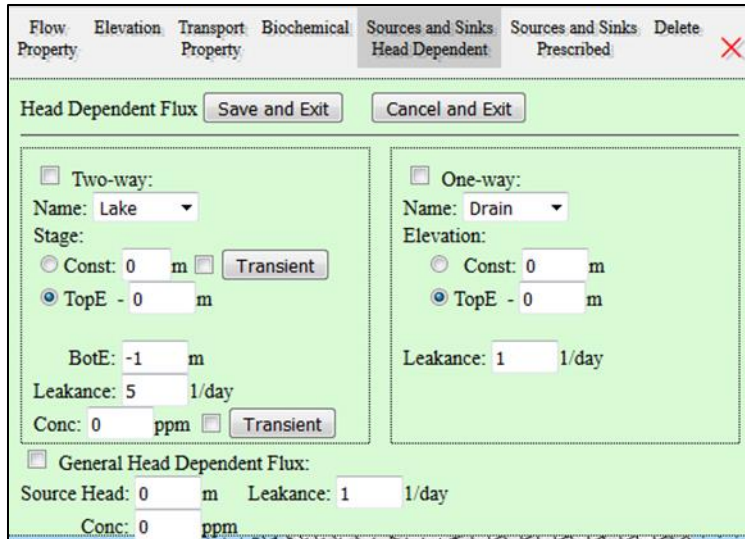


Figure 4-13: Sources and Sinks (Head Dependent) sub-menu (for Zone conceptual features).

If the time-varying option is desired, LM click on ‘Transient’ button to open the ‘Transient Data for Lake Stage’ menu (see Figure 4-14). Indicate the starting data and time with the text boxes next to ‘State Date’ (year, month, day, hour). Then, provide the following inputs for each timestep: TimeN – or the number of days since the starting date and time; and ValueN – the stage value corresponding to TimeN. The stage values may be entered in any units; however, if meters are not used, make sure to enter the appropriate multiplier in the text box next to ‘Multiplier to meter:’. For example, to convert from ft. to m, enter 0.3048, as 1 ft.=0.3048m. After all desired changes have been made, LM click the ‘OK’ button. Or click ‘Cancel’ to discard all changes and close the sub-menu.

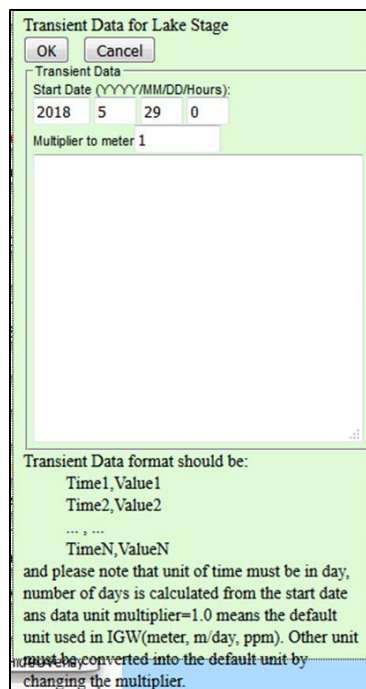


Figure 4-14: Sub-menu for editing transient lake (or other source/sink) stage data.

Use the text box next to 'BotE:' assign a bottom elevation of the source/sink (e.g., the lake bottom elevation). Use the text box next to 'Leakance:' to enter a value (units of day^{-1}) for the leakancy (a factor representing hydraulic conductivity per unit thickness of the land surface).

Two-way source concentration. The two-way source/sink can be assigned a solute concentration for the purposes of transport modeling (in the case that the zone is acting as a source). Enter a non-zero value in the text box next to 'Conc:' (units: ppm) to apply a source concentration to the source/sink.

To assign time-varying source concentration, check the radio box next to the 'Transient' button, and then LM click the 'Transient' button. Check the radio box next to the 'Transient' button to open the Transient Data for Lake Concentration sub-menu (see Figure 4-15). Indicate the starting data and time with the text boxes next to 'Start Date' (year, month, day, hour). Then, provide the following inputs for each timestep: TimeN – or the number of days since the starting date and time; and ValueN – the concentration value corresponding to TimeN. The concentration values may be entered in any units; however, if ppm are not used, make sure to enter the appropriate multiplier in the text box next to 'Multiplier to ppm'. After all desired changes have been made, LM click the 'OK' button. Or click 'Cancel' to discard all changes and close the sub-menu.

See Sections 5.2 and 6.1 for details on visualizing solute/contaminant transport results.

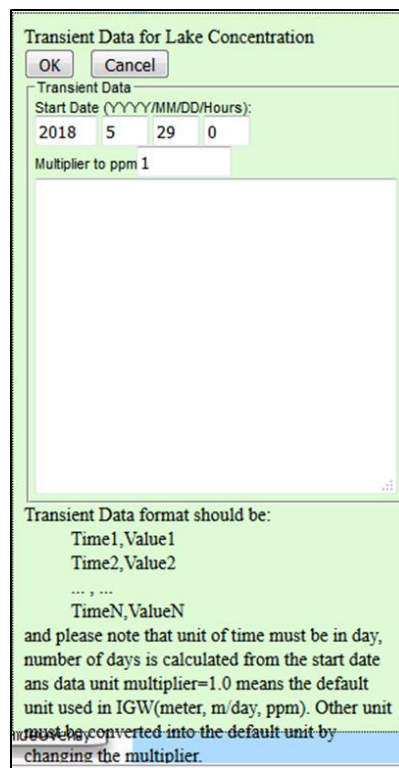


Figure 4-15: Sub-menu for editing transient lake concentration data.

One-way head-dependent flux. One-way head dependent flux allows water to leave the aquifer system in the case that the hydraulic head exceeds the top aquifer elevation, but water may not enter the aquifer system. To assign the zone as a one-way sink of water (e.g., seeps or springs), check the radio box next to 'One-way'. The drop-down menu next to 'Name' provides different naming options for the zone feature that will be used in displaying results, e.g., when performing water balance analyses (see Section 6.2). The stage of the water sink can be assigned as:

- a constant value that does not change with time – select the radio button next to ‘Const:’ and enter the desired value)
- the aquifer top elevation minus some prescribed value; by default, the stage is set to the aquifer top elevation, as indicated by the value of zero in the text box next to ‘Top E – ’ shown in Figure 4-13.

Use the text box next to ‘BotE:’ assign a bottom elevation of the source/sink (e.g., the lake bottom elevation). Use the text box next to ‘Leakance:’ to enter a value (units of day⁻¹) for the leakancy (a factor representing hydraulic conductivity per unit thickness of the land surface).

General Head Dependent flux. Check the box next to “General Head Dependent Flux:’ to assign the zone as a pollutant/solute source. This requires assigning a source head (entered as a single value in the text box next to ‘Source head’), the leakancy (units of day⁻¹), and the source concentration (units of ppm).

4.2.6 Sources and Sinks (Prescribed)

Click on the ‘Sources and Sinks Prescribed’ tab title to access the Prescribed Flux and Sources sub-menu (see Figure 4-16). This sub-menu allows for assigning:

- zone-specific recharge as a prescribed flux, steady or unsteady;
- a prescribed head value, steady or unsteady; or
- a source concentration, either as “instantaneous”, continuous or time-varying.

Figure 4-16: Sources and Sinks (Prescribed) sub-menu (for Zone conceptual features).

Prescribed Recharge. Check the radio box next to ‘Recharge:’ and enter a non-zero value (units: inch/year) in the text box to the right of ‘Recharge:’ to apply a prescribed recharge for all model cells over which the zone extends. This value is steady (i.e., the same value will be applied for all time-steps if performing transient simulations).

To assign time-varying recharge, check the radio box next to the ‘Transient’ button, and then **LM** click the ‘Transient’ button. This opens the ‘Transient Data for Recharge’ sub-menu (see Figure 4-17). Indicate the starting data and time with the text boxes next to ‘State Date’ (year, month, day, hour). Then, provide the following inputs for each timestep: TimeN – or the number of days since the starting date and time; and ValueN – the recharge value corresponding to TimeN. The recharge values may be entered in any units; however, if m/day are not used, make sure to enter the appropriate multiplier in the text box next to ‘Multiplier to m/day:’. For example, to convert from inch/year to m/day, enter 0.00006958, as 1 inch/year = 0.00006958 m/day. After all desired changes have been made, **LM** click the ‘OK’ button. Or click ‘Cancel’ to discard all changes and close the sub-menu.

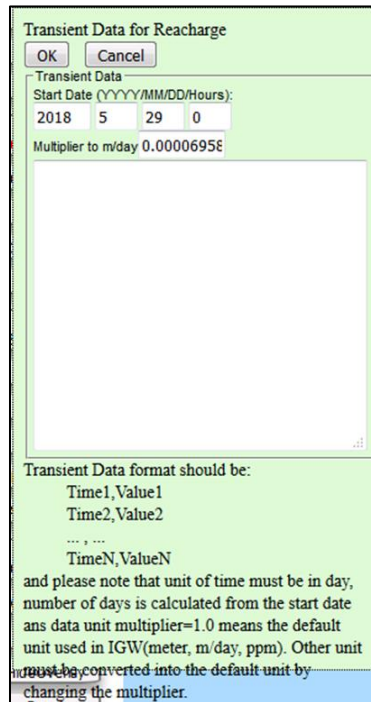


Figure 4-17: Sub-menu for editing transient recharge data.

Prescribed Source Concentration. Check the radio box next to ‘Recharge’ and enter a non-zero value (units: inch/year) in the text box next to ‘Conc:’ to apply a prescribed source concentration (units: ppm) for all model cells over which the zone extends. This value is steady (i.e., the same value will be applied for all time-steps if performing transient simulations).

To assign time-varying source concentration, check the radio box next to the ‘Transient’ button, and then **LM** click the ‘Transient’ button. This opens the ‘Transient Data for Recharge Concentration’ sub-menu (see Figure 4-18). Indicate the starting data and time with the text boxes next to ‘State Date’ (year, month, day, hour). Then, provide the following inputs for each timestep: TimeN – or the number of days since the starting date and time; and ValueN – the concentration value corresponding to TimeN. The concentration values may be entered in any units; however, if ppm are not used, make sure to enter the appropriate multiplier in the text box next to ‘Multiplier to ppm:’. After all desired changes have been made, **LM** click the ‘OK’ button. Or click ‘Cancel’ to discard all changes and close the sub-menu.

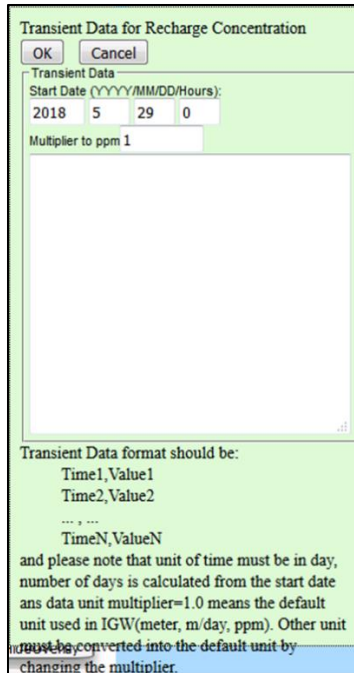


Figure 4-18: Sub-menu for editing transient recharge concentration data.

Prescribed Head. Check the radio box next to ‘Prescribed Head:’ to treat the zone as a prescribed head boundary condition. The head can be assigned as:

- a constant value that does not change with time – select the radio button next to ‘Constant:’ and enter the desired value;
- a constant value that is time-varying – after selecting the radio button next to ‘Constant:’, check the radio box next to the ‘Transient’ button
- the aquifer top elevation minus some prescribed value; by default, the stage is set to the aquifer top elevation, as indicated by the value of zero in the text box next to ‘Top E

To assign time-varying head, check the radio box next to the ‘Transient’ button, and then click the ‘Transient’ button. This opens the ‘Transient Data for Prescribed Head’ sub-menu (see Figure 4-19). Indicate the starting date and time with the text boxes next to ‘Start Date’ (year, month, day, hour). Then, provide the following inputs for each timestep: TimeN – or the number of days since the starting date and time; and ValueN – the head value corresponding to TimeN. The head values may be entered in any units; however, if meters are not used, make sure to enter the appropriate multiplier in the text box next to ‘Multiplier to meter:’. After all desired changes have been made, click the ‘OK’ button. Or click ‘Cancel’ to discard all changes and close the sub-menu.

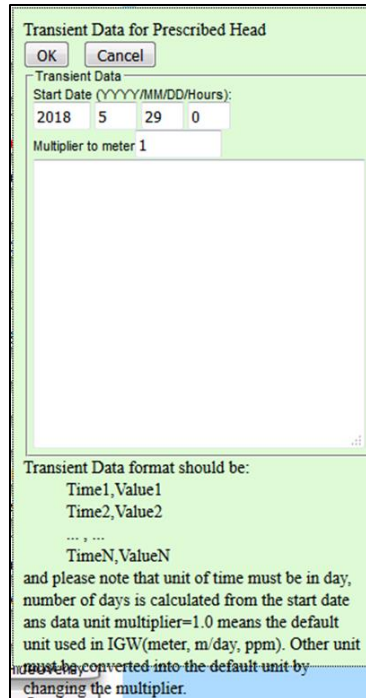


Figure 4-19: Sub-menu for editing transient prescribed head data.

Prescribed Source Concentration. A zone can be assigned a solute source concentration for all model cells over which it extends for the purposes of transport modeling. To do this, check the radio box next to ‘Source Concentration’. To apply an instantaneous (delta-function) source concentration, select the radio button next to ‘Instantaneous:’ and enter a value (units: ppm) in the text box provided. To apply a continuous concentration (i.e. constant in time), select the radio button next to ‘Continuous:’ enter a value (units: ppm) in the text box provided

To assign time-varying source concentration, check the radio box next to the ‘Transient’ button, and then LM click the ‘Transient button Check the radio box next to the ‘Transient’ button to open the Transient Data for Lake Concentration sub-menu (see Figure 4-20). Indicate the starting data and time with the text boxes next to ‘State Date’ (year, month, day, hour). Then, provide the following inputs for each timestep: TimeN – or the number of days since the starting date and time; and ValueN – the concentration value corresponding to TimeN. The concentratoin values may be entered in any units; however, if ppm are not used, make sure to enter the appropriate multiplier in the text box next to ‘Multiplier to ppm’. After all desired changes have been made, LM click the ‘OK’ button. Or click ‘Cancel’ to discard all changes and close the sub-menu.

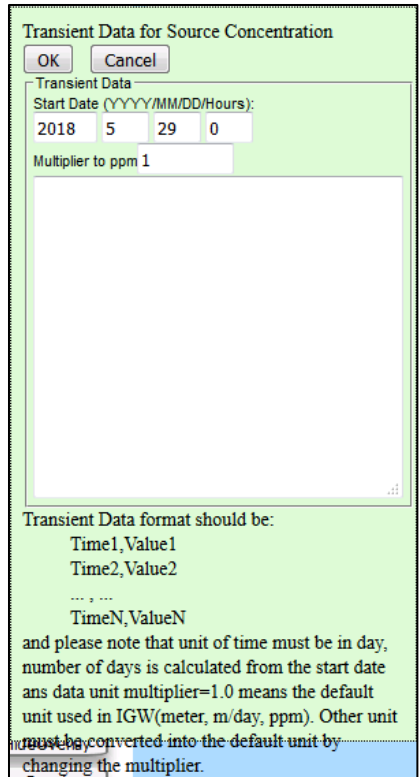
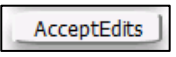
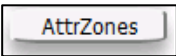


Figure 4-20: Sub-menu for editing transient source concentration data.

4.2.7 *Editing Attributes of Existing Zones*

When the simulation is complete/paused, any pre-existing zone feature can be edited by clicking the 'AttrZones' button from left-side of the MAGNET Modeling Environment. All pre-existing zones are then filled red within the model domain (see Figure 4-21). Click once within the zone to be edited, and select 'OK' from the prompt to confirm editing. This opens the Zones Attributes menu for updating parameters/options of the polyline feature. Click 'Save and Exit' to apply the changes to the zone feature, and continue editing other zone features, if desired. Once all changes have been made, click on the 'AcceptEdits' button on the left-side of the MAGNET Modeling Environment. This applies all updates and turns the zone back to green (unfilled) objects in the MDWA.



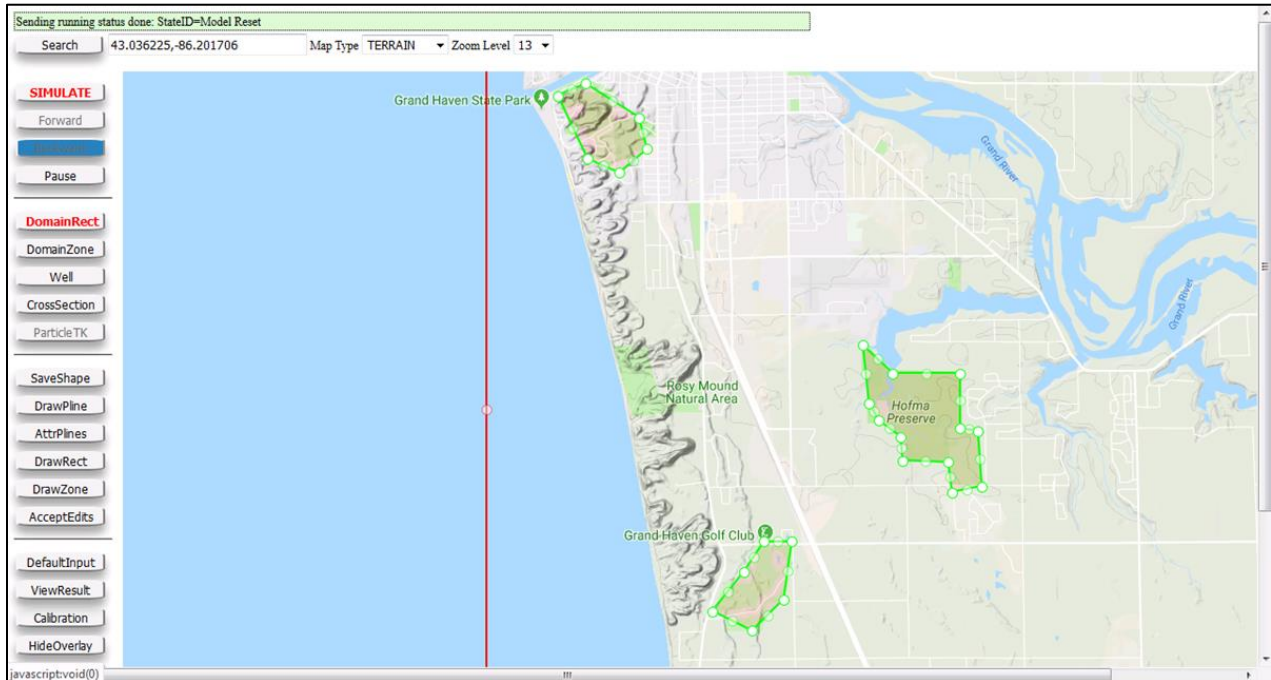


Figure 4-21: attributes of pre-existing zone features.

4.3 Well Features & Attributes

Wells can be added to the model domain to:

- add pumping stress or a source of water (e.g., injection well) at a point location
- add a point-source of a pollutant/solute in groundwater
- place particles at a point location or in a circle around a point location (see Section 5.6)
- monitor simulation results (both head and concentrations) and compare with observations (see Section 6.1).

To add a well feature to the model domain, **LM** click on the ‘Well’ button along the left-side of the MAGNET Modeling Environment (see Section 2.5). The cursor will become a cross-hair, allowing for placement of a well with a single-click of the **LM** button. This will open the Well Input Options menu (see Figure 4-22). The coordinates (latitude and longitude) of the well feature are displayed in degree-decimal form in the text boxes next to ‘Lat:’ and ‘Lon:’.¹³

¹³ The location of any well can be updated by entering new values in these text-boxes, then hitting ‘OK’.

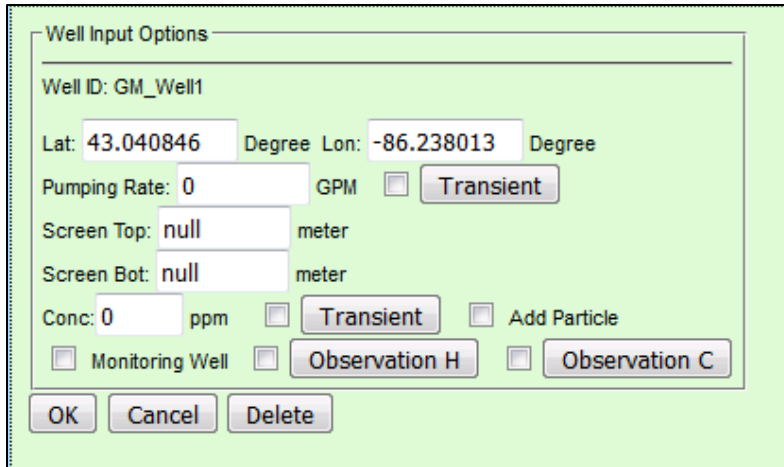
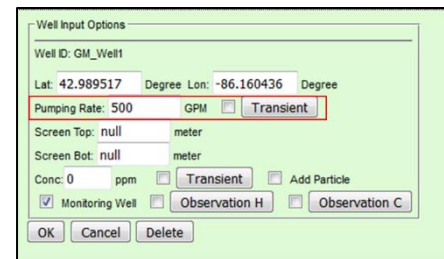


Figure 4-22: Well Input Options menu.

4.3.1 Well Features as Pumping/Injection Wells

By default, the pumping rate is equal to zero. To make the well a sink (pumping stress), enter a negative value in the text box next to ‘Pumping Rate:’, or enter a positive value to conceptualize it as an injection well. The value that is entered is used for steady-state pumping (i.e., the rate does not change with time).

The default location of the well screen will be centered at a distance from the surface that is two-thirds the depth of the aquifer at that location. The user may specify the screen top elevation and bottom elevation by entering non-zero values (units: meters) in the text boxes next to ‘Screen Top:’ and ‘Screen Bot:’, respectively.



To implement a time-varying pumping/injection rate, check the radio box next to ‘Transient’, then LM click on the ‘Transient’ button. This opens the Transient Data for Pumping/Injection Rate sub-menu (see Figure 4-23). Indicate the starting data and time with the text boxes next to ‘State Date’ (year, month, day, hour). Then, provide the following inputs for each timestep: TimeN – or the number of days since the starting date and time; and ValueN – the pumping rate value corresponding to TimeN. The pumping rates may be entered in any units; however, if m³/day are not used, make sure to enter the appropriate multiplier in the text box next to ‘Multiplier to m³/day:’. After all desired changes have been made, LM click the ‘OK’ button. Or click ‘Cancel’ to discard all changes and close the sub-menu.

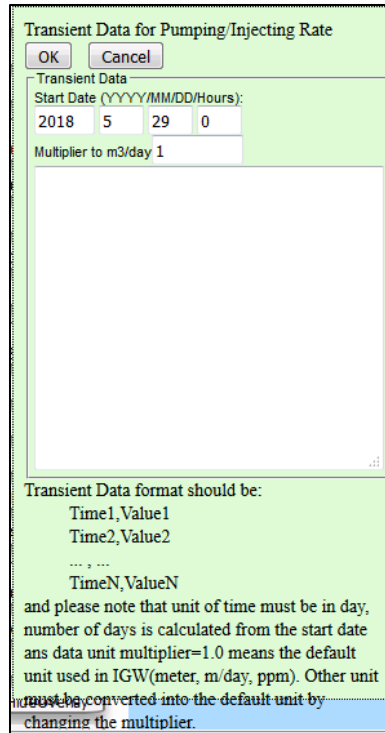
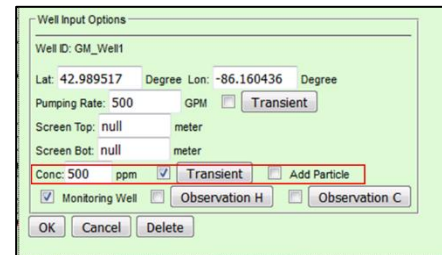


Figure 4-23: Sub-menu for editing transient pumping/injection rate data.

4.3.2 Well Features as a Point Source

To use a well feature as a point source (e.g., an injection well) of pollutants/solutes, enter a non-zero number in the text box next to 'Conc:' (units: ppm)¹⁴. The value that is entered is used as source concentration for all time steps (i.e., the well functions as a continuous source). See Sections 5.2 and 6.1 for details on visualizing solute/contaminant transport results.



To implement a time-varying source concentration, check the radio box next to the 'Transient' button, then LM click on the 'Transient' button. This opens the Transient Data for Injecting Source Conc Rate sub-menu (see Figure 4-24). Indicate the starting data and time with the text boxes next to 'Start Date' (year, month, day, hour). Then, provide the following inputs for each timestep: TimeN – or the number of days since the starting date and time; and ValueN – the concentration value corresponding to TimeN. The concentrations may be entered in any units; however, if ppm are not used, make sure to enter the appropriate multiplier in the text box next to 'Multiplier to ppm'. After all desired changes have been made, LM click the 'OK' button. Or click 'Cancel' to discard all changes and close the sub-menu.

¹⁴ A positive pumping rate must be entered in the text box next to 'Pumping Rate' for the well to function as a point source. The resulting mass flux of the pollutant/solute from the point source is: (pumping rate)*(concentration).

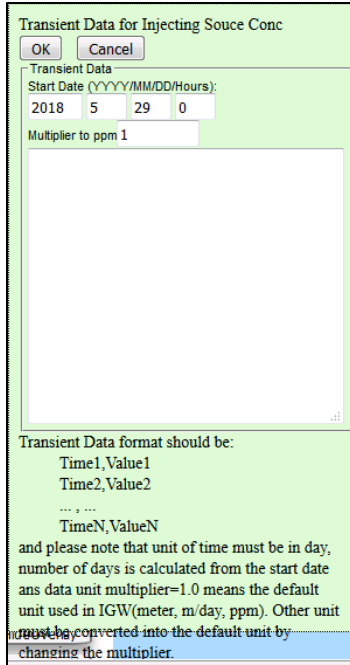


Figure 4-24: Sub-menu for editing transient injection source concentration data.

Placing Particle at a Well Location. Adding particles around wells and then performing backward particle tracking is a very useful way to delineate well capture zones. To add particles around a well, check the radio box next to 'Add Particle'. The number of particles placed around a well can be controlled accessing the Default Parameters and Options menu by clicking on the 'DefaultInput' button along the left-side of the Magnet Modeling Environment (see Section 2.7). In the Particle Tracking option box, adjust the number of particles around a well by entering a value in the text box next to 'Particle # Around a Well:' (see Figure 4-25). The particles will be placed a radial distance from the well location determined by the value (units: meters) entered in the text box next to 'Particle Well Radius:'. See Section 5.6 for complete details on the particle tracking capabilities and options available in MAGNET.

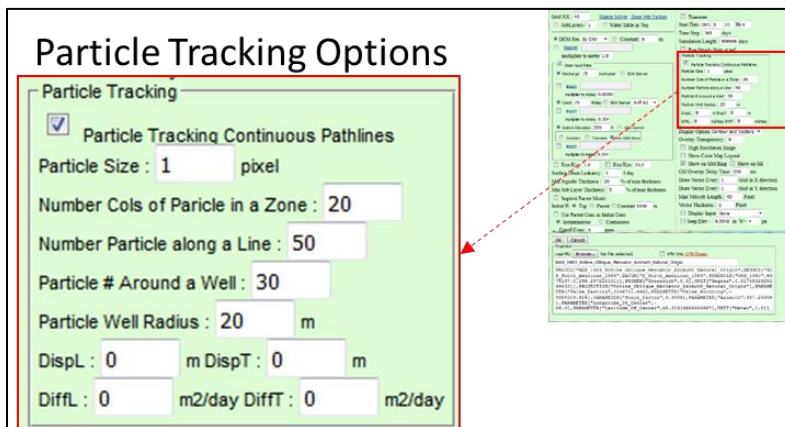
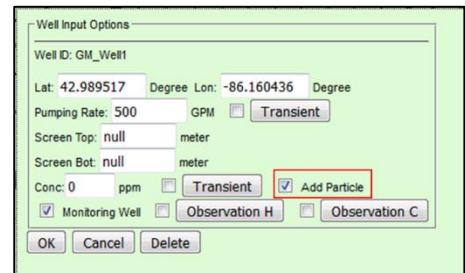


Figure 4-25: Particle tracking options available in the Default Parameters and Options menu. Default values/settings are shown.

The Well Input Options can also create a monitoring well at the location of the placed well feature. To do so, check the radio box next to ‘Monitoring Well’. When the model is simulated/run forward, a time-series plot for head and/or concentration will be available in the Monitoring Wells Plot chart when analyzing model results with the ‘ViewResult’ button (see Section 6.1).

Including Observational Data. To include transient head and/or solute concentration observations in the monitoring well plots, check the radio boxes next to ‘Observation H’ and ‘Observation C’ button.

To input transient head data, LM click on the ‘Observation H’ button. This opens the Transient Data for Head Observation sub-menu (see Figure 4-26). Indicate the starting data and time with the text boxes next to ‘State Date’ (year, month, day, hour). Then, provide the following inputs for each timestep: TimeN – or the number of days since the starting date and time; and ValueN – the head value corresponding to TimeN. The head values may be entered in any units; however, if meters are not used, make sure to enter the appropriate multiplier in the text box next to ‘Multiplier to meter’. After all desired changes have been made, LM click the ‘OK’ button. Or click ‘Cancel’ to discard all changes and close the sub-menu.

To input concentration data, LM click on the ‘Observation H’ button. This opens the Transient Data for Head Observation sub-menu (see Figure 4-27). Indicate the starting data and time with the text boxes next to ‘State Date’ (year, month, day, hour). Then, provide the following inputs for each timestep: TimeN – or the number of days since the starting date and time; and ValueN – the concentration value corresponding to TimeN. The concentrations may be entered in any units; however, if ppm are not used, make sure to enter the appropriate multiplier in the text box next to ‘Multiplier to ppm’. After all desired changes have been made, LM click the ‘OK’ button. Or click ‘Cancel’ to discard all changes and close the sub-menu.

Transient Data for Head Observation

OK Cancel

Transient Data

Start Date (YYYY/MM/DD/Hours):

2018 5 29 0

Multiplier to meter 1

Transient Data format should be:

Time1,Value1
Time2,Value2
...
TimeN,ValueN

and please note that unit of time must be in day,
number of days is calculated from the start date
and data unit multiplier=1.0 means the default
unit used in IGW(meter, m/day, ppm). Other unit
must be converted into the default unit by
changing the multiplier.

Figure 4-26: Sub-menu for editing transient head observations.

Transient Data for Conc Observation

OK Cancel

Transient Data

Start Date (YYYY/MM/DD/Hours):

2018 5 29 0

Multiplier to ppm 1

Transient Data format should be:

Time1,Value1
Time2,Value2
...
TimeN,ValueN

and please note that unit of time must be in day,
number of days is calculated from the start date
and data unit multiplier=1.0 means the default
unit used in IGW(meter, m/day, ppm). Other unit
must be converted into the default unit by
changing the multiplier.

Figure 4-27: Sub-menu for editing transient concentration observations.

After well inputs have been finalized, the well feature will appear as a yellow circle in the model domain (see Figure 4-28). To edit or delete any existing well feature, hover the cursor over it and **LM** click. This brings up the Well Input Options for that feature. Edit as desired, then **LM** click 'OK'. Or delete the well feature by **LM** clicking on the 'Delete' button.

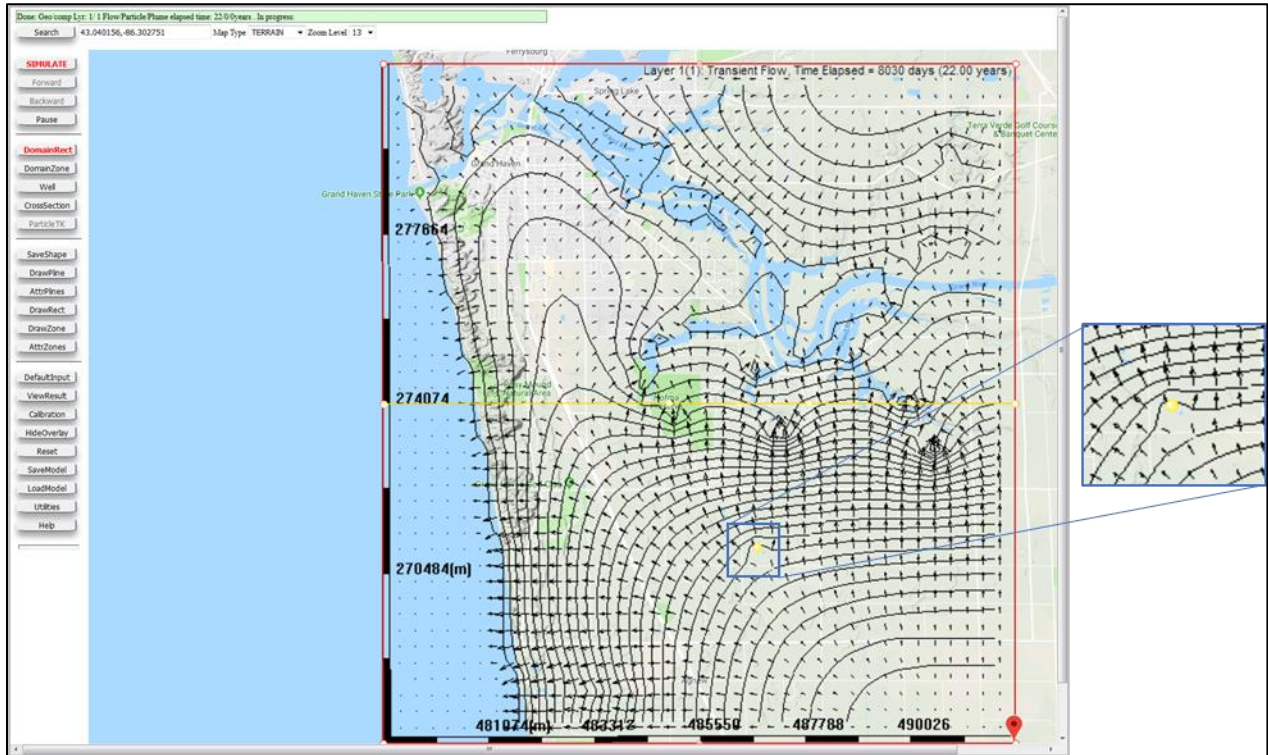


Figure 4-28: Example of a well feature placed in a model.

4.4 Geometry Features in Bing Maps Version

MAGNET allows users to use either Google Maps or Bing Maps as the background environment. There are slight differences in how a user adds and edits geometric features (model domain, zones/polygons, polylines) between the two. The preceding sections are written for the Google Maps based version. The following subsections explain how to add/edit geometric features when using the Bing Maps based version.

4.4.1 Adding Geometric Features

- A. Add a well (point location) --- left-mouse (LM) click anywhere in the maps display.
- B. Add a rectangle --- click-drag the LM button from the first vertex (node) location to the vertex location diagonal from the first, then release the LM button to end the shape.
- C. Add a polyline --- LM click in the map display to form the polyline (i.e., create nodes and line segments), then, double click to end the shape.
- D. Add a zone ---LM click at map to form a polygon (i.e., add nodes and perimeter line segments), then double click to end the shape.

4.4.2 Editing Geometric Features

Edit Method 1 – using the mouse device

- A. Click the right mouse (RM) click at (or as close as possible to) a polyline node – then click-drag the selected node to edit the geometry. If there is no node movement, move the cursor closer to the polyline and repeat the click-drag.
- B. RM click inside (or along the boundary of) a rectangular shape; you will be asked to continue to edit the shape or not. To edit: answer OK, then click-drag one of the 4 nodes to edit the geometry.
- C. RM click inside (or boundary of) a zone shape, then drag the node to edit the geometry.
- D. If the AttrPlines or AttrZones button is selected (button title will become 'acceptEdits'), RM clicking a attribute Pline or attribute Zone will launch the attribute form for you to edit.
- E. Once you are done, click 'SaveShape' to save the edits.

Edit Method 2 -- mobile device and mouse device

- A. LM click/tap 'editShape', then LM click/tap the domain polygon, a conceptual polygon (zone), a particle polygon, a Cross section, or a particle Pline (polyline) to edit the geometry.
- B. LM click/tap AttrPlines first, then LM click 'editShape', then LM click/tap the polyline of interest to edit both geometry and attributes.

C. LM click/tap AttrZones first, then LM click/tap editShape, then LM click/tap the polygon of interest to edit both geometry and attributes.

D. if AttrPlines or AttrZones is clicked (button title will become 'acceptEdits'), then method A will not work - you have to 'acceptEdits' before you do A

E. LM click/tap 'SaveShape' to save the edits.

In Edit Method 2, the process of LM clicking/tapping the 'editShape' button and then selecting a geometry feature functions like a RM click directly on a shape when using desktop machine. Therefore, the 'editShape' button must be clicked first when you want to edit a new geometry. This means you have to click this button every time before you click at/inside a new geometry feature. In short, a RM is equal to LM clicking/tapping the 'editShape' button then LM click/tap the shape/feature of interest.

For an example, when you are trying to select a polyline:

a. Click editShape

b. Move your cursor on or as close as to the polyline, then LM click/tap.

If you fail to select the geometry because of the inaccuracy of the LM click/tap, then you have repeat a and b until you are successful.

Chapter 5 SIMULATION AND VISUALIZATION

As described in Chapter 1, MAGNET is designed so that any model can be simulated with results provided in “real-time” (i.e., nearly instantaneously if a modest number of grid cells and conceptual features are used). The results can be used to guide model editing or to build new, nested models for more detailed local analysis (see subsection 3.6.1). This chapter explains how to simulate and run a model and view the model results.

5.1 Submitting a Model for Simulation

Once the model domain has been created, a user may submit the model (or “job”) for simulation by clicking on the red ‘SIMULATE’ button on the left-side of the MDWA (see section 2.4). The first time a job is submitted, the user must provide their account credentials, i.e., username and password (see Section 1.3). All subsequent job submissions during the web session will not require entering user credentials.



Note that users need not change any default inputs/options or add any conceptual features in order to run a model. This “one-click” steady-state modeling approach allows users to quickly see the large-scale, prevailing flow patterns that are largely controlled by the topographic framework (see Curtis et al., *in preparation*), which is by default set as the top aquifer elevation surface (see subsection 3.3.1).

Once the model has been submitted for simulation, MAGNET will automatically search for the appropriate geographic projection (‘PRJ’) to use based on the region used for modeling. A prompt will appear asking the user to accept the projection (LM click the ‘OK’ button) or use the default projection (click the ‘Cancel’ button). See the following sub-section for details on defining the default projection.



5.1.1 Defining the Default Projection System

To define a default projection system to use for modeling, navigate to the Default Parameters and Options menu by LM clicking on the ‘DefaultInput’ button along the left-side of the MAGNET Modeling Environment (see Section 2.7). LM click the ‘Browse’ button next to ‘Load PRJ’ in the Projection option box (see Figure 5-1). This will open a window to browse to a file for uploading. After the projection system file is loaded, the system details will be shown in the large text window below the name of the current default projection system¹⁵.

¹⁵ By default, the NAD_1983_NSRS2007_Michigan_GeoRef_Meters (State of Michigan) projection system is used in MAGNET.

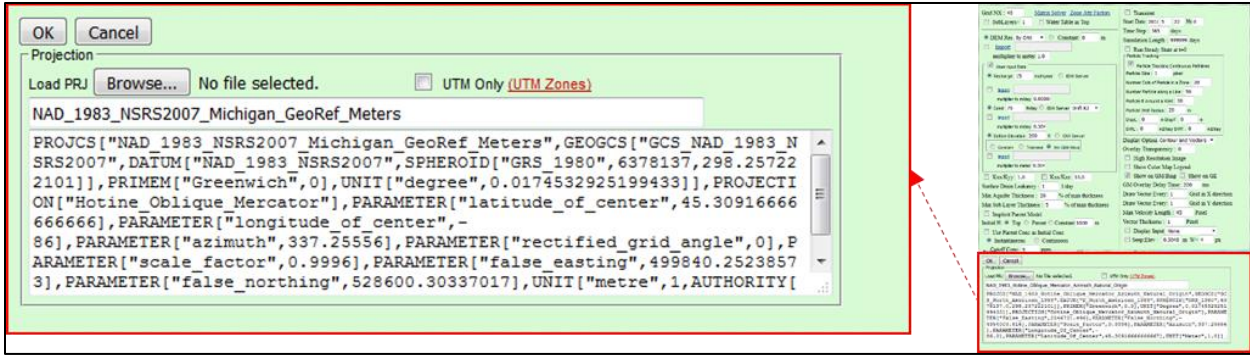


Figure 5-1: Projection option box in the Default Parameters and Options menu.

5.1.2 Simulation Status

After the PRJ system is selected, the Status Bar (see Section 2.2) will provide information on the current status of the model submitted for simulation (see Table 5-1 for examples). Pay attention to the progress of the model. If there is an issue with the simulation process, an error message will be shown (see the last row in Table 5-1). Sometimes simply re-submitted the model for simulation will alleviate the issues (e.g., if there is an issue reaching the IGW server). In other cases, editing/refinement if input data/files may be necessary (e.g., if the format for user input file is not correct).

Table 5-1: Examples of Status Bar messages displayed while submitting a model for simulation

PRJ found: NAD_1983_NSRS2007_Michigan_GeoRef_Meters
Connecting to IGW server:#3(1 to 5) and running the model, Please wait ..
Running on IGW server...
Running on IGW server...: Drawing velocity arrows
Running on IGW server...: Converting Image File ...in PRJ1
Running on IGW server...: Layer 2-->mapping 'Drain' to grid points ...
Done: Geo/comp Lyr: 1/ 1 Flow/Particle Plume elapsed time: 6/0/0years . In progress:
Error: no available IGW server, try agin later.

After the head/solute distribution is solved for the initial time-step, the Status Bar will read “Done: Geo/comp In progress”. The resulting head (and solute distribution) will be shown within the model display in the MDWA (see Figure 5-2 for an example). A cross-section, shown as a yellow polyline spanning the entire model from west-east (W-E) along the central portion of the domain, is automatically generated for visualizing vertical characteristics of the aquifer system (see Section 5.4). Customizing the display of results is explored in the next section.

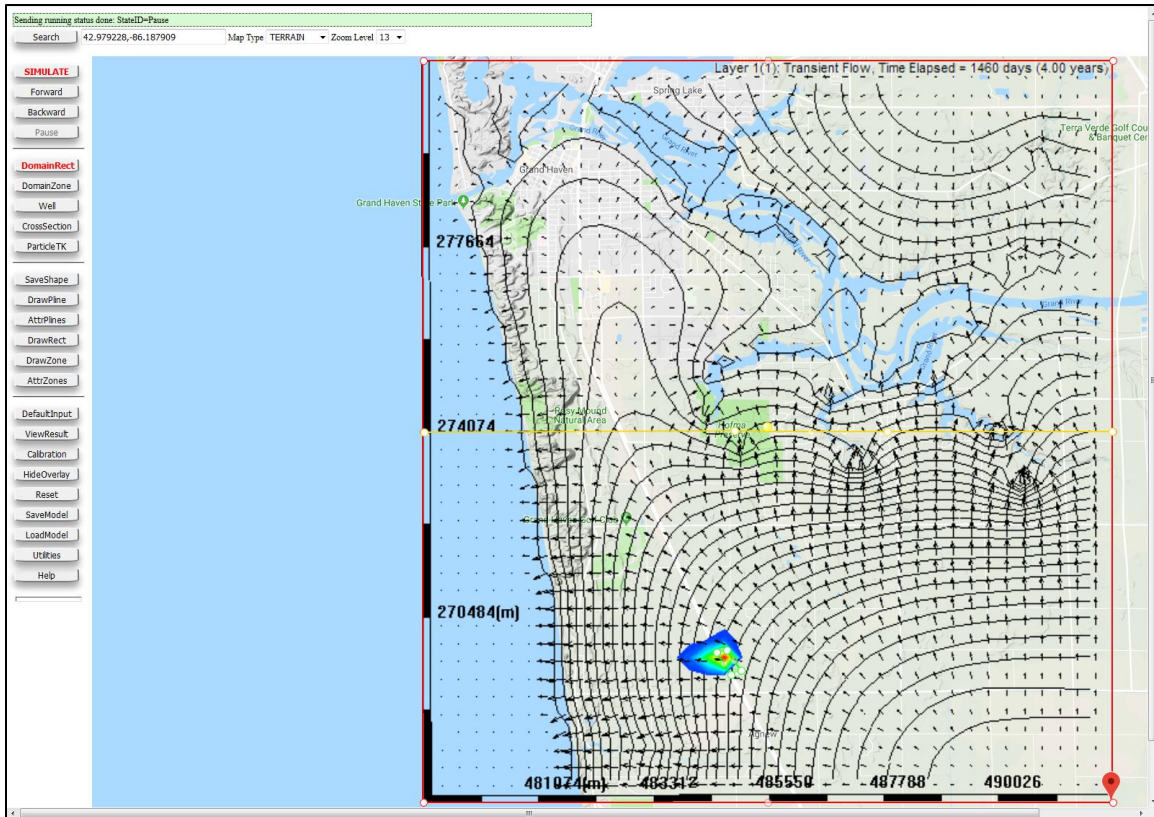
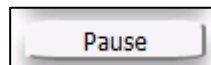


Figure 5-2: Example of simulation results (head and concentration). Note that the solute concentration is indicated with the color overlay (red = high concentration; blue = low concentration).

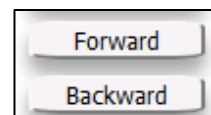
5.1.3 Pausing, Editing, and Submitting New Models

Pausing a Model. At any point in the simulation, users can pause it by LM clicking on the ‘Pause’ button along the left-side of the MAGNET Modeling Environment (see Section 2.4). MAGNET will finish its calculations and the model redraws for the present time step before appearing idle. Once the simulation is paused, the user may:



- Add particles for particle tracking (note that this will reset the particle tracking clock – see Section 5.6).
- View model results and perform analyses (see Sections 5.3-5.6 and Chapter 6)
- Update display options (see Section 5.2)
- Hide/show the model results, or reset./save the model (see Chapter 7)

After making adjustments, the user can resume simulation by clicking on the ‘Forward’ button along the life-side of the MDWA. The option to run the simulation in reverse (e.g., for the purposes of reverse particle tracking), LM click on the ‘Backward’ button instead.



Editing and Submitting a New Model. Any changes to the model conceptualization (e.g., conceptual features added or updated, the aquifer elevations are assigned differently, etc.) require that the model is re-submitting for simulation. Once the red ‘SIMULATE’ is selected, a prompt will appear asking the user to

confirm a RESET of the currently running model. Select ‘OK’ to implement the changes to the model conceptualization. If ‘Cancel’ is selected, an option to load the currently running model is provided.

If a model domain of a different size is desired (e.g., in the case of hierarchical modeling – see Section 3.6), use ‘DomainRect’ or ‘DomainZone’ to draw the new model domain (see Section 3.1). Note that the results of the old model will be lost, although they may still be utilized for assigning boundary conditions of the new model (see Section 3.6).

5.2 Display Options

MAGNET provides the user with a variety of options concerning the visualization of modeling parameters and results in the Working Area. This section details those options.

Figure 5-3 presents the display options available in the Default Model Parameters and Options menu.

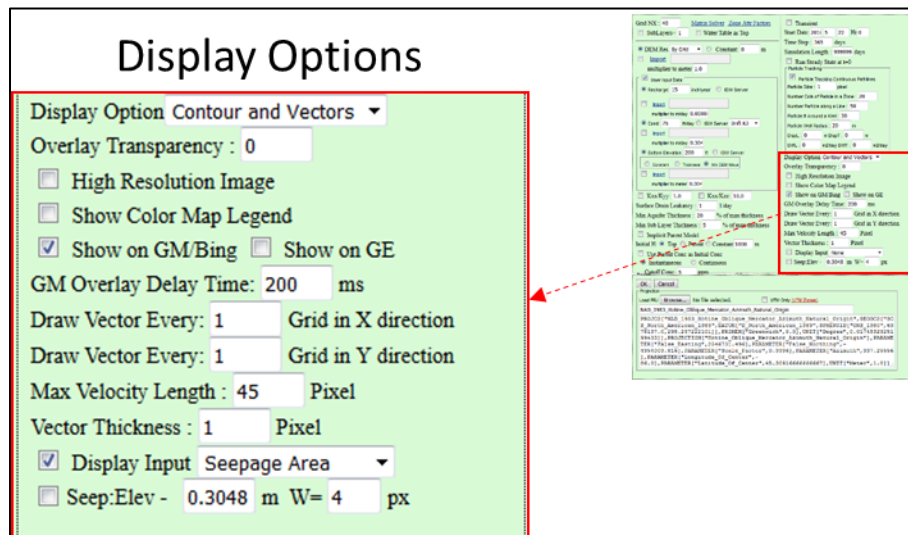
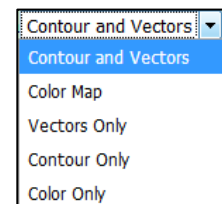


Figure 5-3: Display options available in MAGNET.

5.2.1 Options for Displaying Flow Results

Use the ‘Display Option’ pull-down menu to choose how the groundwater flow results will be displayed:

- with black head contour lines and velocity vectors
- with a head color map overlaid by black contour lines and velocity vectors
- with velocity vectors only
- with head contours only
- and with a color map only



Examples of each are shown in Figure 5-4. The text box next to ‘Overlay Transparency:’ determines the transparency level (0-100%) of the groundwater flow results overlay. By default, this is set to zero.

When using the color map for display, check the radio box next to ‘Show Color Map Legend’ to add a legend to the top-left portion of the model domain. Units of head are meters in MAGNET.

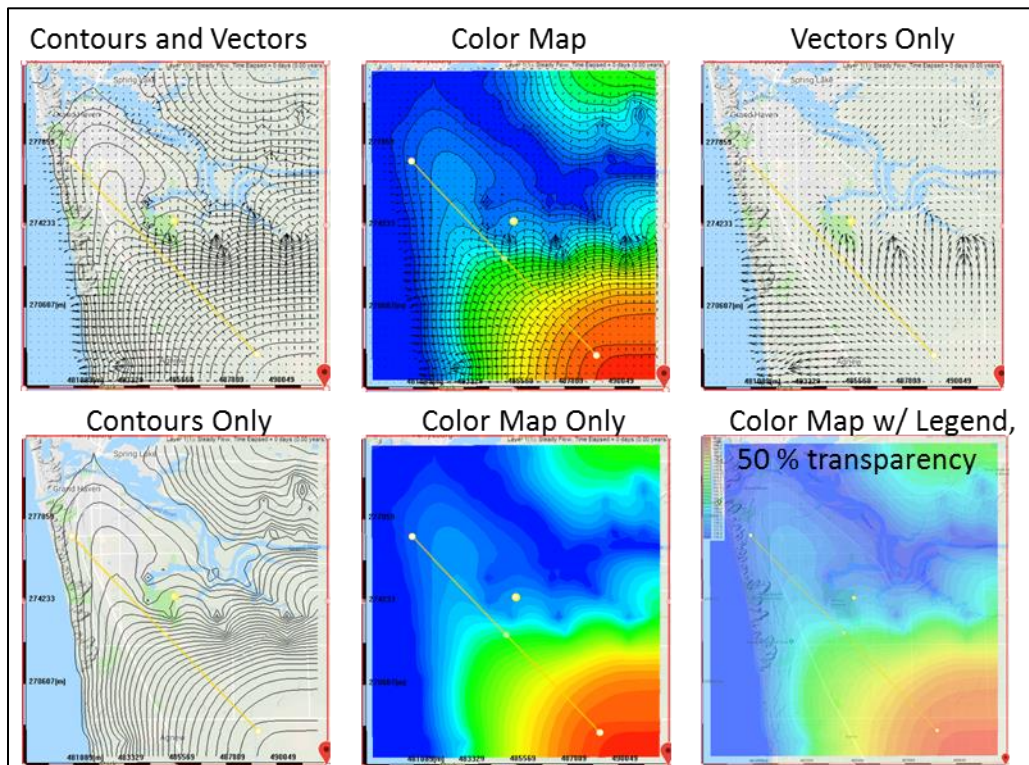


Figure 5-4: Different ways to display the groundwater flow results (head distribution and seepage velocities).

Vector Characteristics. The density of velocity vectors shown in the display is determined by the number grid cells to skip before drawing another velocity vector, both in the X (W-E) and Y (N-S) directions. Use the text boxes provided next to ‘Draw Vector Every:’ to specify how often a velocity vector should be drawn in the display. By default, vectors are drawn every grid point in both the X and Y directions.

Draw Vector Every: 1	Grid in X direction
Draw Vector Every: 1	Grid in Y direction

The length of the vectors is controlled by specifying the maximum velocity vector length (in terms of display pixels).

Vector Thickness : 1	Pixel
----------------------	-------

5.2.2 Options for Displaying Model Inputs

In addition to displaying model results, MAGNET allows for plan-view visualization of model inputs. This includes:

- hydraulic conductivity
- top and bottom aquifer elevations
- recharge
- land use/land cover
- soil type
- aquifer thickness
- transmissivity

To display a model input, check the radio box next to 'Display Input', then select the model variable from the corresponding drop-down menu (see Figure 5-5). Click 'OK' to save the changes, then re-submit the model for simulation¹⁶. The model inputs will appear in the model domain when the simulation is finished.

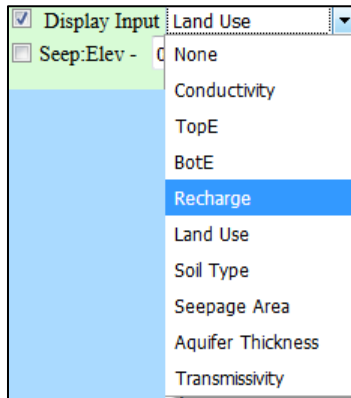


Figure 5-5: Drop-down menu used to select an model input for display.

Examples of model inputs being displayed within the model domain are shown in Figure 5-6

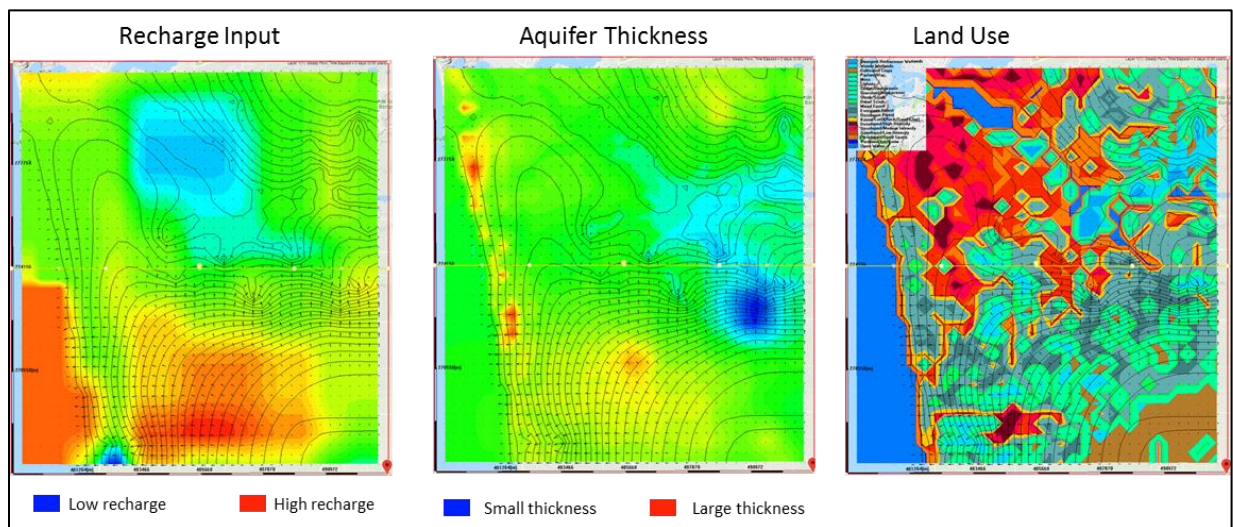


Figure 5-6: Examples of displaying model inputs in the MDWA.

5.2.3 *Displaying Simulated Seepage Area*

One of the options in the 'Display Input' drop-down menu is 'Seepage Area'. When this is selected and the radio box next to 'Display Input' is checked, the updated display will include purple cross-hair graphics in locations where groundwater is seeping to the surface (i.e., water is leaving the aquifer to discharge to a surface water body such as a stream, lake, wetland, etc.). An example is shown in Figure 5-7.

¹⁶ It is only necessary to re-submit the model for simulation when performing steady-state modeling. If performing transient modeling, simply pause the model, update the display input option, then run the model forward. The updated display input option will appear after the following simulation time-step is solved.

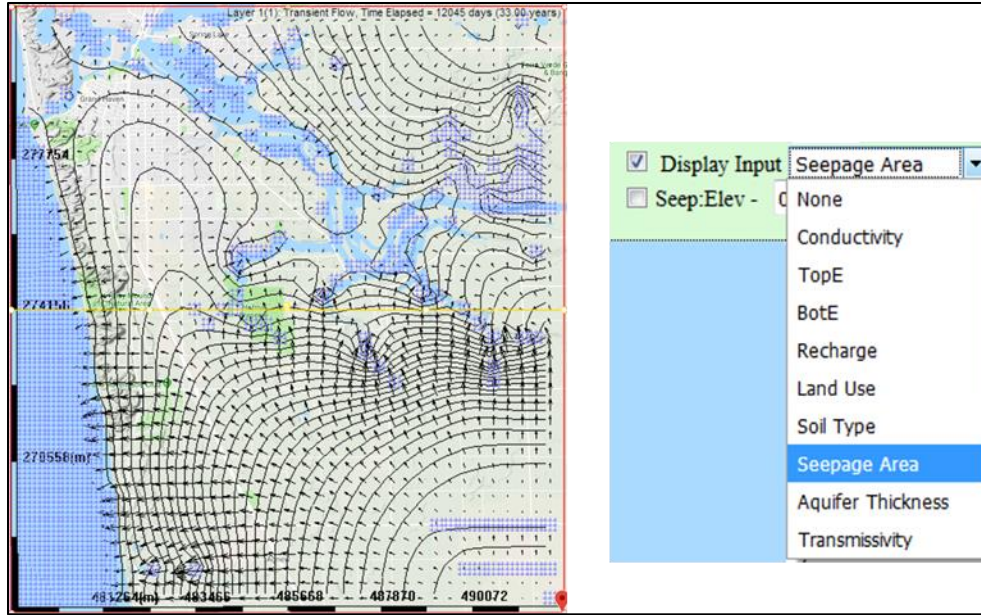
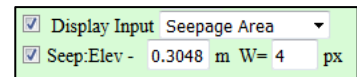


Figure 5-7: Example of using the ‘Seepage Area’ option from the ‘Display Input’ drop-down menu to indicate areas where groundwater is discharging at the land surface.

By default, the land surface is used as the top elevation for computing seepage (in the case where the hydraulic head is larger than the top elevation). However, an offset can be applied to determine where the hydraulic head is higher than some distance below the land surface. For example, one might be interested in knowing where the water table intersects the root zone (about 1 ft. below the surface) across the model domain.

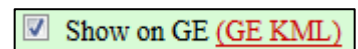
To apply an offset, check the radio box next to ‘Seep: Elev -’, and enter the offset amount (units: meters) in the provided text box. The ‘W=’ text box is used to specify the length (units: pixels, or px) of each cross-hair used to depict a grid cell in which seepage is occurring.



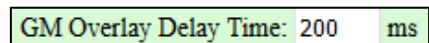
5.2.4 Exporting to Google Maps/Earth

Checking the radio box next to ‘high resolution image sends back a double-resolution image of current model output¹⁷.

To view the model inputs and simulation results in Google Earth, check the radio box next to ‘Show on GE’. After the model simulation is completed, a link, ‘(GE KML)’, will be available in the Default Model Parameters and Options menu. Click on the link to download the .kml file to be loaded in Google Earth. An example kml file opening in Google Earth is shown in Figure 5-8.



The value in the text box next to ‘GM Overlay Delay Time:’ is the delay time (units: milliseconds) to be used for displaying updated model results in Google Earth.



¹⁷ This feature is being phased out of the next version of MAGNET.

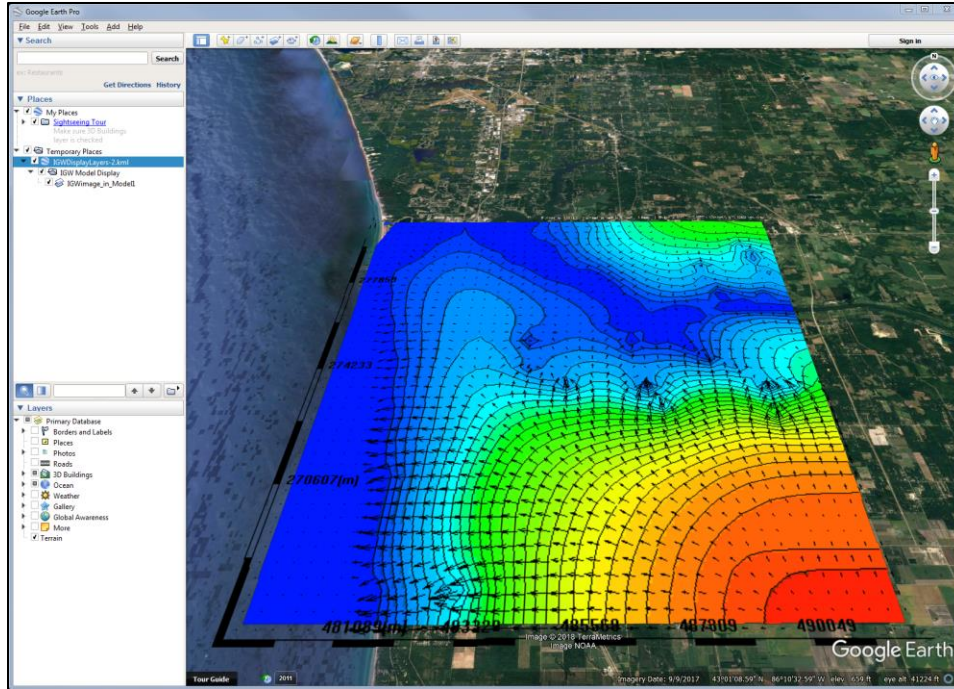


Figure 5-8: Example of displaying flow model results in Google Earth.

5.2.5 Hiding and Re-Displaying Simulation Results

When the simulation is completed/paused, the plan-view simulation results can be hidden from the display using the ‘HideOverlay’ in the lower-left side of the MDWA (see Section 2.7). All simulation results (e.g., head contours, velocity vectors, particle paths, concentration colormap, etc.). The model domain (shown in red), cross-section polyline (shown in yellow and the conceptual features (e.g., zone extents and polylines) will remain in the MDWA (see Figure 5-9).

HideOverlay

After the ‘HideOverlay’ button is clicked, it will become the ‘ShowOverlay’ button, which can be used to re-display the simulation results.

ShowOverlay

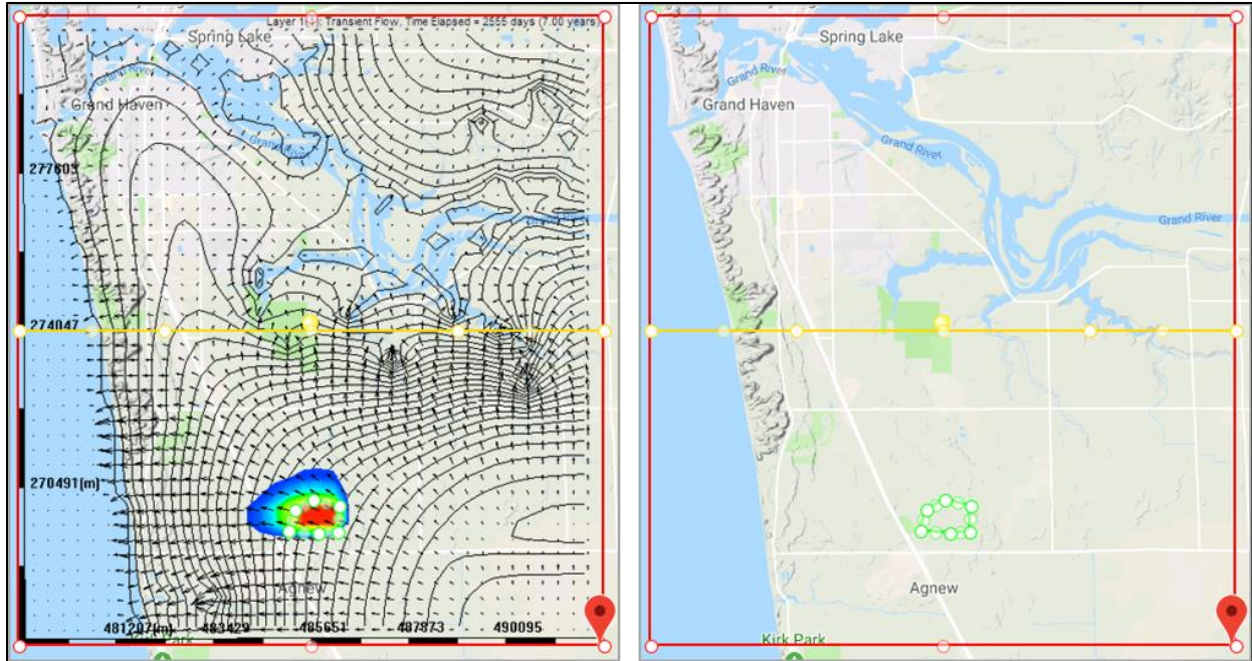


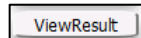
Figure 5-9: Example of using the 'HideOverlay' button to temporarily remove the simulation results from the MDWA.

5.3 Model Values at a Node

At any point during a simulation, the user may add 'nodal value flags' within the model domain to display variable values in the model at for the current time step¹⁸. The variables include:

- grid location (I,J,K) of the nearest node and the geological and computational layer
- simulation time that has elapsed
- simulated hydraulic head
- hydraulic conductivity
- recharge
- layer top and bottom elevations
- seepage velocity in the W-E direction (V_x), N-S direction (V_y), and vertical direction
- solute concentration
- soil type
- land use/land cover

To add nodal value flags, LM click on the 'ViewResult' button along the left-side of the MAGNET Modeling Environment (see Section 2.7), then select 'Nodal Values' from the pop-up menu. Then single LM click anywhere within the model domain to place a nodal value flag. The cursor will continue to place nodal value flags with single LM clicks within the model domain.



¹⁸ Add a nodal value flag to the model will automatically pause the simulation.

Once a flag is placed, hover the cursor over it and single **LM** click to view the variable values of the nearest node. An example is shown in Figure 5-10. Close the variable table for an individual flag by clicking on the grey 'X' in the right-hand corner of the table.

To remove the flags and return the cursor to “normal” mode, **LM** click on the ‘ViewResult’ button, then select ‘NodalV Off’.

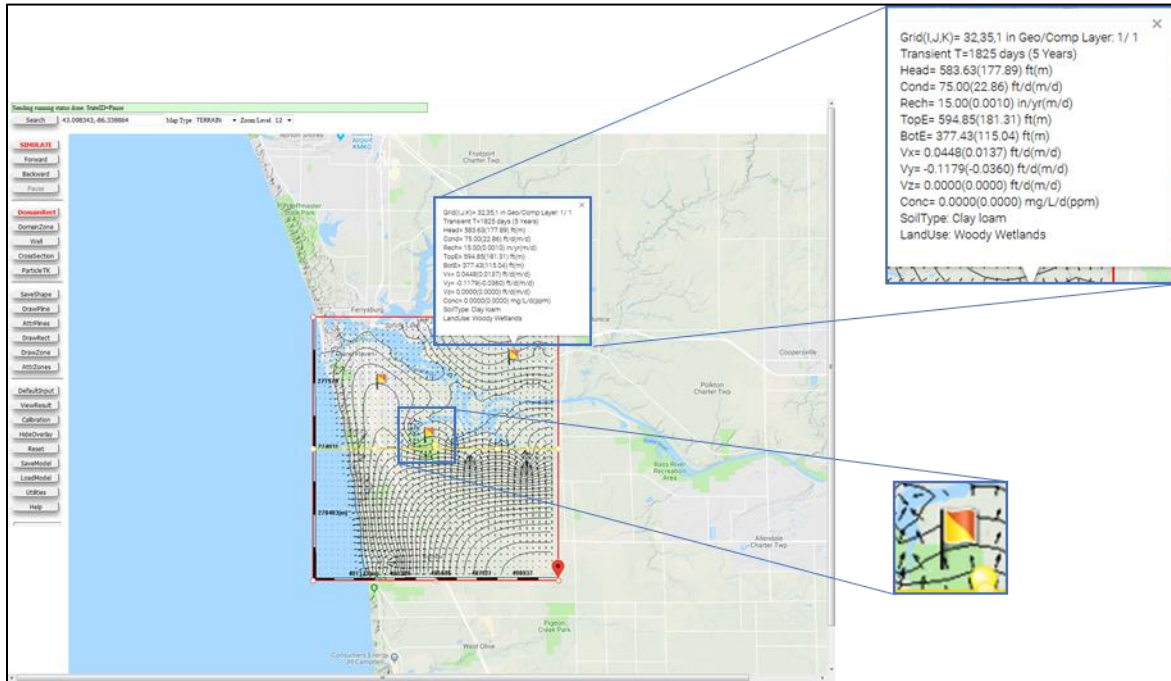


Figure 5-10: Example of placing nodal value flags in the model domain, with a sample variable table shown for the flag in the upper right of the model domain.

5.4 Cross-sections

As mentioned in subsection 5.1.2, a cross-section is automatically generated for visualizing vertical characteristics of the aquifer system. The location and extent of the cross-section, represented as a yellow polyline, can be adjust by hovering the mouse over any of the four vertices and click-dragging to the new desired location.

To view the model results along the currently placed cross-section, **LM** click on the ‘ViewResult’ button along the left-side of the MAGNET Modeling Environment, then select ‘Display Charts’ from the pop-up menu. This launches five different charts:

- Monitoring Wells plot (see Section 6.1)
- Cross-section plot
- Cross-section diagram (“Full Cross-section”)
- Mass Balance Bar Chart (see Section 6.2)
- 3D surface plot (see Section 5.5)

Any of the charts can be moved by click-dragging the chart frame across the MDWA. Options available for modified plots can be accessed by clicking on the pair of red arrows at the bottom-left of the chart.

An example of charts generated when using the ‘Display Charts’ option in the ‘ViewResults’ pop-up menu is shown in Figure 5-11, with the example Cross-section plot and Cross-section diagram highlighted/enlarged for emphasis. The next two subsections discuss each charter in further detail.

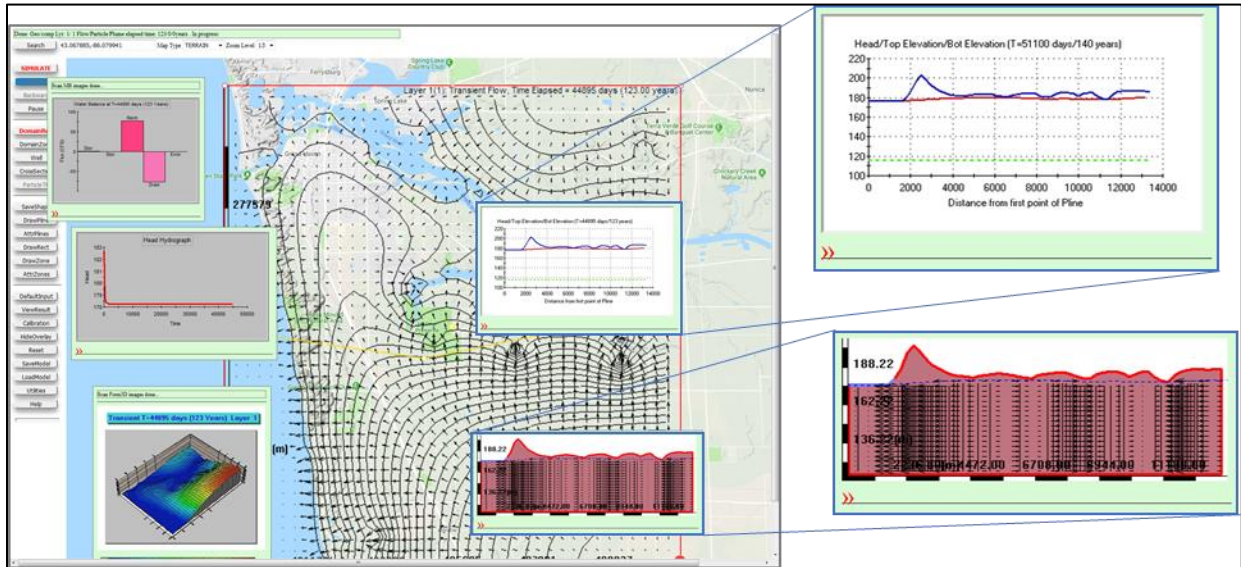


Figure 5-11: Example charts generated when using the ‘Display Charts’. An example Cross-section plot is shown in the top-right, and an example Cross-section diagram (Full cross-section) is shown in the bottom-right.

5.4.1 Cross-section Plots

The Cross-section Plot is used to graphically depict variations of head, concentration, conductivity, aquifer top and bottom elevations, and/or seepage velocities (V_x and V_y) along the direction of the cross-section. The plot can be customized by accessing the plot option (expand the red arrows at the bottom-left of the chart – see above). Illustrative examples are shown in Figure 5-12. By default, results for head, top elevation and bottom elevation are shown for the top-most layer (Figure 5-12a).

Use the radio boxes in the scroll menu to determine which variables to display in the Cross-section Plot. Check the radio box next to ‘Legend’ to add a plot legend on the right-side of the chart. Click on the arrows next to ‘Layer #=’ to change layers (if the ‘Show Single Layer:’ radio button is selected), or select the radio button next to ‘Show All Layers’ to include results for each model layer on the plot.

After making all desired changes, click on the ‘ReDraw’ button to update the plot.

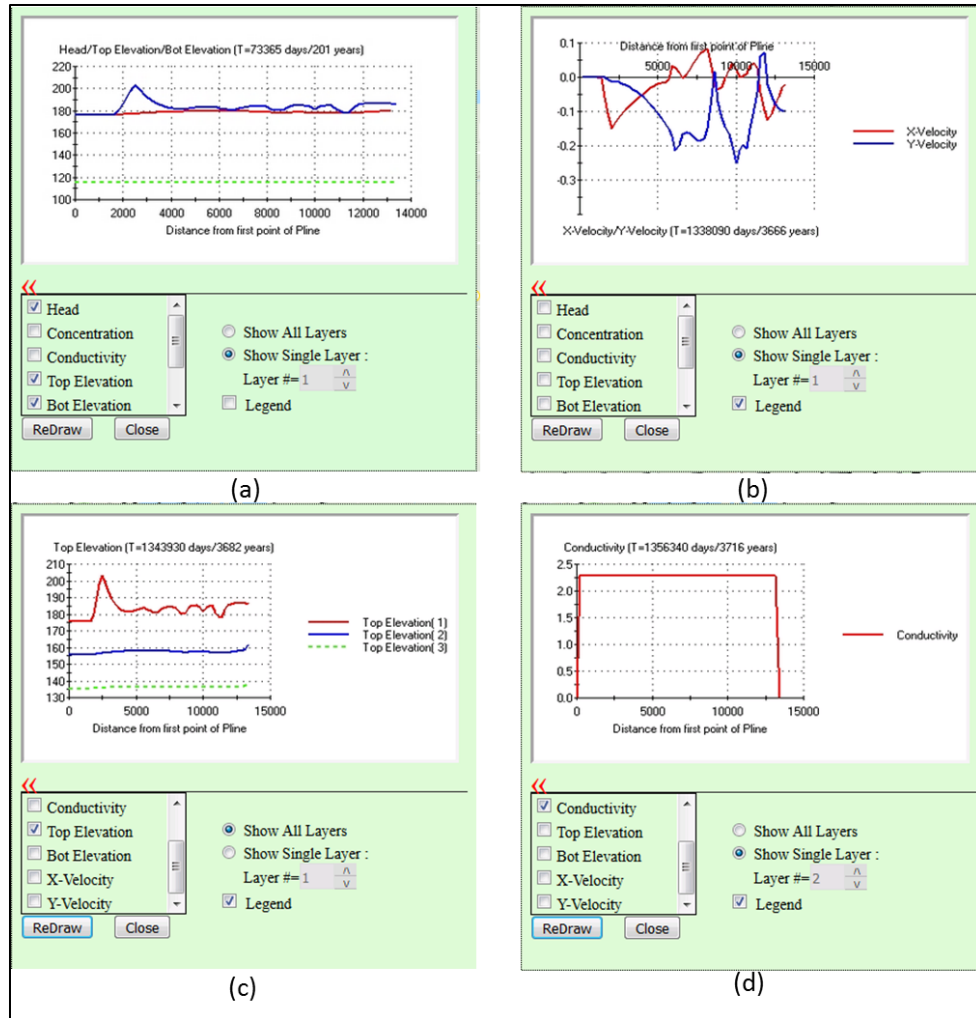


Figure 5-12: Different configurations of the Cross-section plot for the Cross-section shown in Figure 5-11.

5.4.2 Cross-section Diagrams

Cross-section Diagrams show modeling results such as solute concentration, velocity vectors, static head contours and static water line (blue dotted line at the top). Computational layers are separated by dotted lines (yellow dotted lines between the layers). The active computational layer in the cross section is bounded by a thick red line. An example is shown in Figure 5-13.

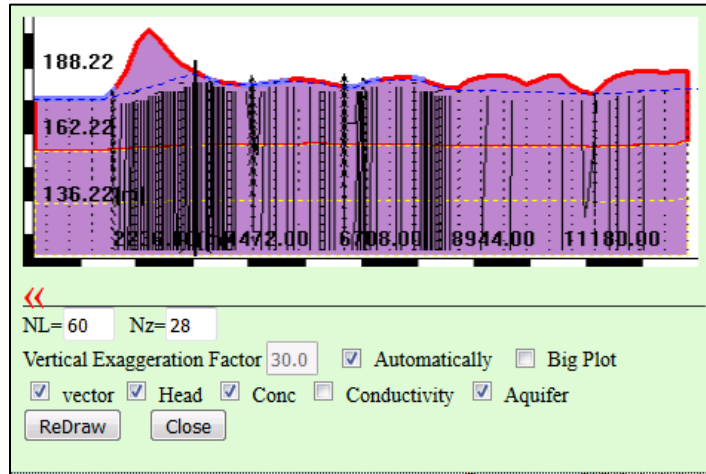


Figure 5-13: Example Cross-section Diagram. Default options/settings are shown.

Users can change the vertical exaggeration factor of the cross-section by unchecking the radio box next to ‘Automatically’, and then single-clicking in the text-box next to ‘Vertical Exaggeration Factor’ to enter a different value. The plot can be enlarged by checking the radio box next to ‘Big Plot’ (see Figure 5-14

After making all desired changes, click on the ‘ReDraw’ button to update the plot.

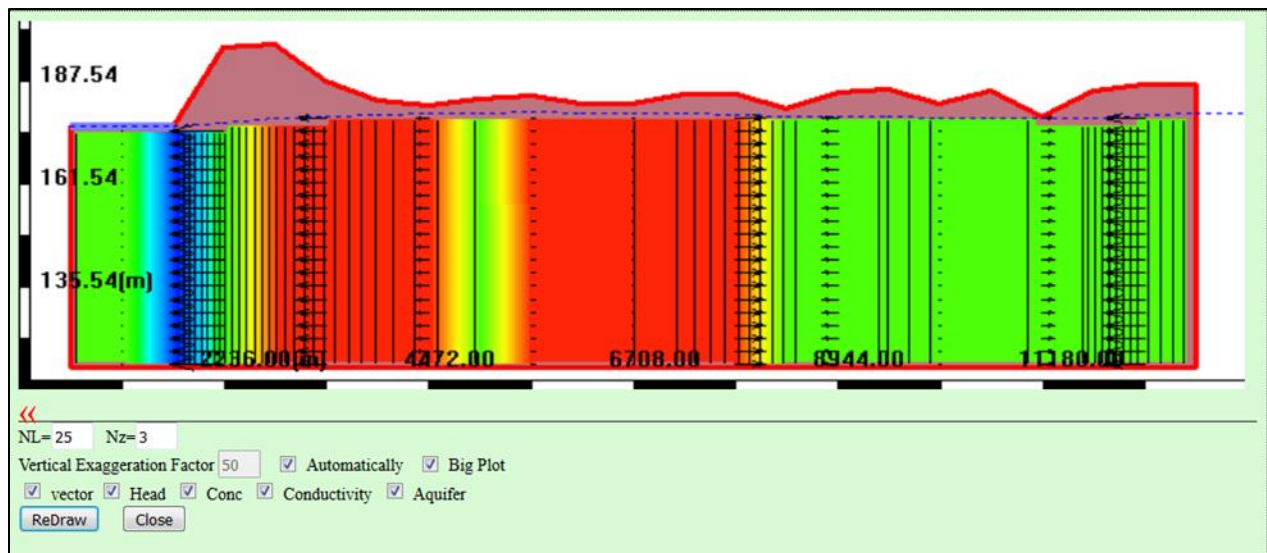


Figure 5-14: Customized Cross-section Diagram. The hydraulic conductivity (K) field is shown, with red colors indicating high K and blue colors indicating low K.

5.5 3D Visualizations

MAGNET allows the model parameters and simulation results to be viewed as a three-dimensional (3D) surface. An example of the chart interface is shown below in Figure 5-15.

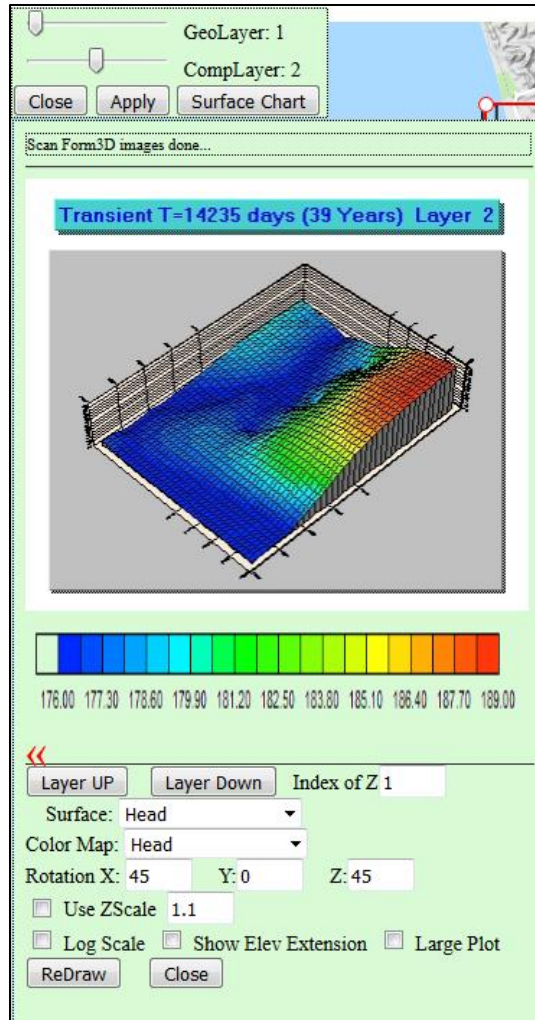


Figure 5-15: Example 3D visualization of a model in MAGNET.

If multiple layers exist in the model, use the scroll bars next to ‘GeoLayer:’ and ‘CompLayer:’ to change geological and/or computational layers, respectively¹⁹. Then click the ‘Surface Chart’ button to display the 3D model visualization. At any point, users can change the geological and/or computational layer and click the ‘Apply’ button to update the chart display. The layer being displayed in the chart can also be updated by using the ‘Layer UP’ and ‘Layer Down’ buttons available in the options menu (accessed by clicking on the red arrows – see above). The number in the text box next to ‘Index of Z’ indicates the layer of the model – ‘1’ corresponds to the bottommost sub/computational layer in a MAGNET model, while the largest number corresponds to the sub/computational topmost layer in a MAGNET model²⁰.

Use the drop-down menu next to ‘Surface’ to change the model variable being displayed. The options to choose from are shown in Figure 5-16. The color map used in the display can also be chosen from the list

¹⁹ In the Basic version of MAGNET, the number of geological/conceptual layers is restricted to one, and the number of computational layers is restricted to three. No such restrictions apply for the Premium version of MAGNET.

²⁰ When multiple geological/conceptual are present (Premium Version only), the numbering scheme is the opposite: ‘1’ corresponds to the topmost conceptual layer in a MAGNET model, while the largest number corresponds to the bottommost layer in a MAGNET model

of variables shown in Figure 5-16. A legend is automatically generated based on the color map variable chosen. Table 5-2 shows the units used for each model variable available for display in the 3D Surface Plot.

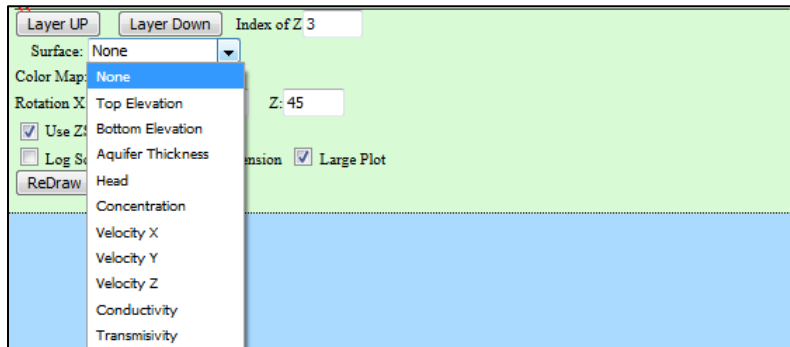


Figure 5-16: Model variables available for display in the 3D Surface Plot.

Table 5-2: Units of model variables displayed in the 3D Surface Plot.

Variable	Units
Top Elevation	m
Bottom Elevation	m
Aquifer Thickness	m
Head	m
Concentration	ppm
Velocity X, Y, Z	m/d
Conductivity	m/d
Transmissivity	m ² /d

Use the text boxes next to ‘Rotation X:’, ‘Y:’, and ‘Z:’ to rotate the display about the X- (W-E), Y- (N-S) and Z- (vertical) axes. To adjust the vertical exaggeration factor, check the radio box next to ‘Use Zscale’, and enter the factor in the provided text-box.

Check the ‘Log Scale’ radio box to display the model variable as log(variable), e.g., when displaying conductivity or transmissivity, which can vary many several orders of magnitude. Check the ‘Large Plot’ radio box to enlarge the chart display. By checking the ‘Show Elev Extension’ radio box, the extents of different geological/conceptual layers (Premium Version Only) will be showed along the sides of the 3D head distribution.

After making all desired changes, click on the ‘ReDraw’ button to update the 3D Surface plot.

5.6 Particle Tracking

A useful way to visualize groundwater flow is to perform traditional particle tracking, in which particles undergoing only advection are introduced at various locations in the flow model, and path lines and travel times are simulated and visualized. Particle tracking allows delineation of path lines in a flow model, which be used, e.g., to determine well capture zones, source water areas, and groundwater delivery mechanisms.

This section details the various ways options for placing and displacing particles during model development and simulation. The four options for placing particles includes:

- placing particles around a well feature (see subsection 4.3.2)
- drawing a rectangle filled with evenly-space particles
- drawing a polygon with evenly-spaced particles
- placing particles along a polyline

The ‘ParticleTK’ button along the left-side of the MAGNET Modeling Environment (see Section 2.5) provides access to the tools need to place particles. Click on it to bring up the Particle pop-up menu (see Figure 5-17). Each tool is discussed in the following subsections.

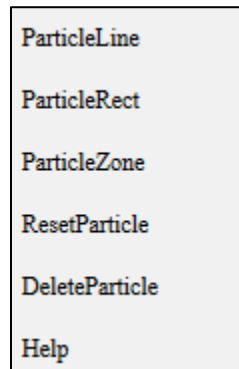


Figure 5-17: Particle Pop-up menu.

As mentioned in subsection 4.3.2, options for displaying particles are available in the Default Model Parameters and Options menu.

5.6.1 *Particles along a Polyline*

Select ‘ParticleLine’ from the Particle pop-up menu to place particles along a polyline. The cursor will become a cross-hair, allowing for placement of polyline vertices with single-clicks of the **LM** button. The polyline will appear as a series of bright yellow line segments with circles at the vertex locations (see Figure 5-18). The location can of any vertex can be changed by hovering the mouse over it and click-dragging to the new desired location. Once the last vertex has been placed, LM click the ‘SaveShape’ button to finalize the polyline.

Run the model forward to perform forward particle tracking, or run the model backward for reverse particle tracking (see subsection 5.1.3 for details on running the model forward or backward). Note that if a particle feature is already present in the model before submitting for simulation, a prompt will help to determine if forward or backward particle tracking will be performed (see right).

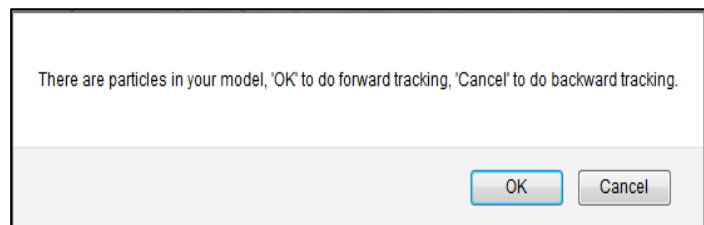


Figure 5-18 provides an example of forward particle tracking of particles placed along a polyline in the bottom-right portion of the model domain. In this example, the path line of the particles is shown, which is the default setting for MAGNET. To remove particle pathlines (and instead only show the particle position at a given time), uncheck the radio box next to ‘Particle Tracking Continuous Pathlines’ in the Default Model Parameters and Options menu. Also note that this example utilizes a default particle size of one pixel. To change the particle size, enter a different value in the text box next to ‘Particle Size :’.

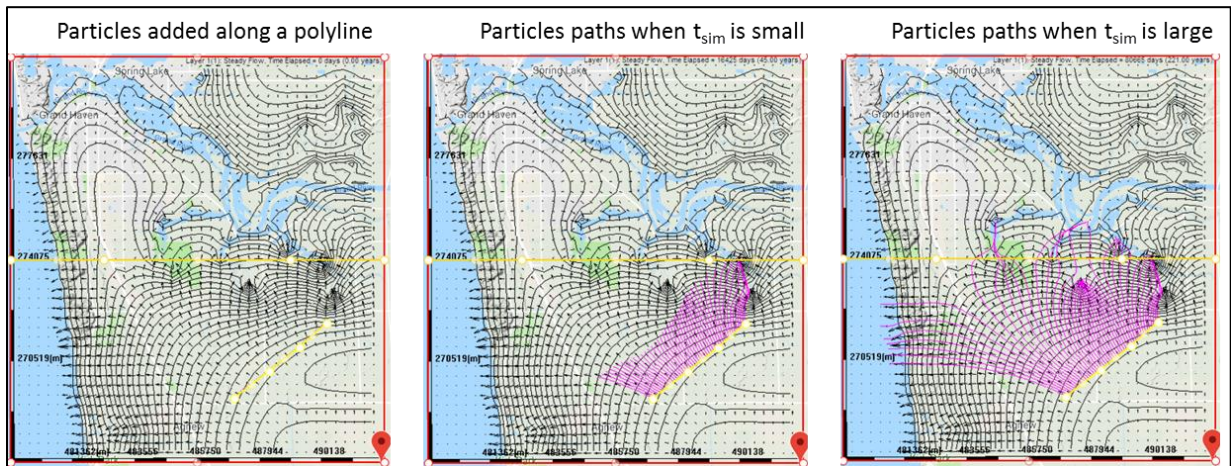
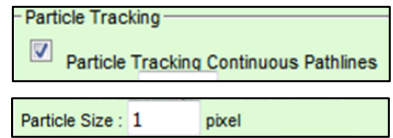


Figure 5-18: Example of placing particles along a polyline and performing forward particle tracking.

The number of particles that are placed (evenly-spaced) along the polyline is controlled by the value in the text box next to ‘Number Particle along a Line:’ in the Default Model Parameters and Options menu.



Drawing a new Particle Polyline. The current version of MAGNET allows for only one particle polyline to be present at one time. If a particle polyline already exists in the model domain, selecting the ‘ParticleLine’ tool from the Particle pop-up menu will automatically remove the existing polyline, and the cursor will once more become a cross-hair for placing the new particle polyline.

5.6.2 Particles in a Zone

Select ‘ParticleRect’ (rectangle) or ‘ParticleZone’ (polygon) from the Particle pop-up menu to place particles inside of a zone feature. The cursor will become a cross-hair, allowing for placement of rectangle/polygon vertices with single-clicks of the **LM** button. The zone will appear as a series of bright yellow line segments with circles at the vertex locations (see Figure 5-19). The location can of any vertex can be changed by hovering the mouse over it and click-dragging to the new desired location. Once the last vertex has been placed, LM click the ‘SaveShape’ button to finalize the particle zone.

Figure 5-19 provides an example of forward particle tracking of particles placed in a zone in the central portion of the model domain. In this example, the path line of the particles is shown, which is the default setting for MAGNET.

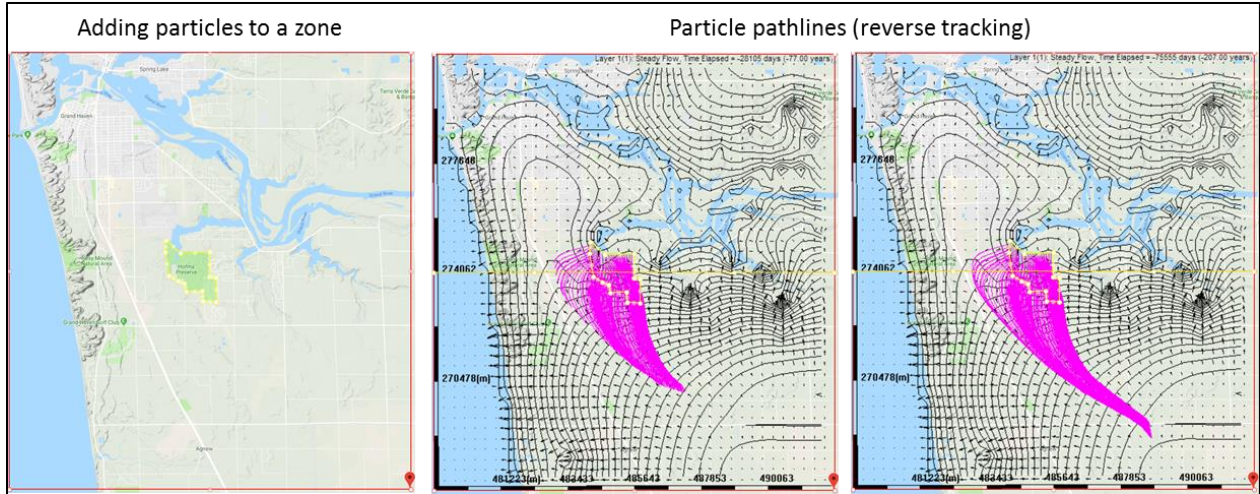


Figure 5-19: Example of placing particles in a zone and performing reverse/backward particle tracking.

The number of particles that are placed (evenly-spaced) within the zone is controlled by the value in the text box next to ‘Number Cols of Particle in a Zone :’ in the Default Model Parameters and Options menu. The value entered represents the number of columns of particles to use to “fill” the zone.

Number Cols of Particle in a Zone : 20

Drawing a new Particle Zone. The current version of MAGNET allows for only one particle zone to be present at one time. If a particle zone already exists in the model domain, selecting the ‘ParticleRect’ or ‘ParticleZone’ tool from the Particle pop-up menu will automatically remove the existing particle zone, and the cursor will once more become a cross-hair for placing the new particle zone.

5.6.3 Adding Dispersion Effects

Dispersion can be incorporated into the particle tracking method in MAGNET. Enter a non-zero value (units: m^2/day) in the text box next to ‘DiffL’ and/or ‘DiffT’ to include lateral (L) and transverse (T) molecular diffusion in the particle tracking simulations. Similarly enter a non-zero value in the text box next to ‘DispL’ and/or ‘DispT’ (units: m) to include lateral (L) and transverse (T) local dispersion in the particle tracking simulations.

DispL : 0 m DispT : 0 m
DiffL : 0 m²/day DiffT : 0 m²/day

5.6.4 Deleting and Resetting Particles

To delete all existing particles from the model, select ‘DeleteParticle’ from the Particle pop-up menu.

DeleteParticle

To reset particles to their initial position (“reset the particle clock”), select ‘ResetParticle’ from the Particle pop-up menu.

ResetParticle

Chapter 6 ANALYSIS & MODEL PERFORMANCE

MAGNET offers several tools for real-time analysis of model results and comparison with observations. In particular, users can:

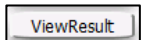
- Monitor head and concentration simulation outputs and compare with time-series measurements at a discrete location.
- Compute water balances for a zone or set of zones
- Compare head measurements across space, either within the entire modeling domain or for a subregion of the domain.

Each of these is explained in further detail in the following sub-sections.

6.1 Monitoring Well Time-Series and Breakthrough Curves

Monitoring wells are useful for visualizing time-series head and concentration data at a discrete location in the model domain. Subsection 4.3.3 provided details on how to make any well feature a monitoring well.

As discussed in Section 5.4, to view Charts of model results, LM click on the ‘ViewResult’ button along the left-side of the MAGNET Modeling Environment, then select ‘Display Charts’ from the pop-up menu. This launches five different charts, including the Monitoring Well plot..



An example of a using a monitoring well to generate a concentration breakthrough curve is shown in Figure 6-1. Note that the ‘Time’ axis is in units of days since the start of the simulation, and the ‘Concentration’ axis is in units of ppm.

An example of using a monitoring well to visualize time-series simulated head data is shown in Figure 6-2. Again, the ‘Time’ axis is in units of days since the start of the simulation. Head is in units of meters.

The Monitoring well plot only shows the results of one monitoring well feature at a time. To show the results for a different feature, use the drop down menu next to ‘Select MW:’ to select a different feature, then click ‘ReDraw’ to update the plot.

Observations of head or concentration can be added to the Monitoring Well plot. See subsection 4.3.3 for more details.

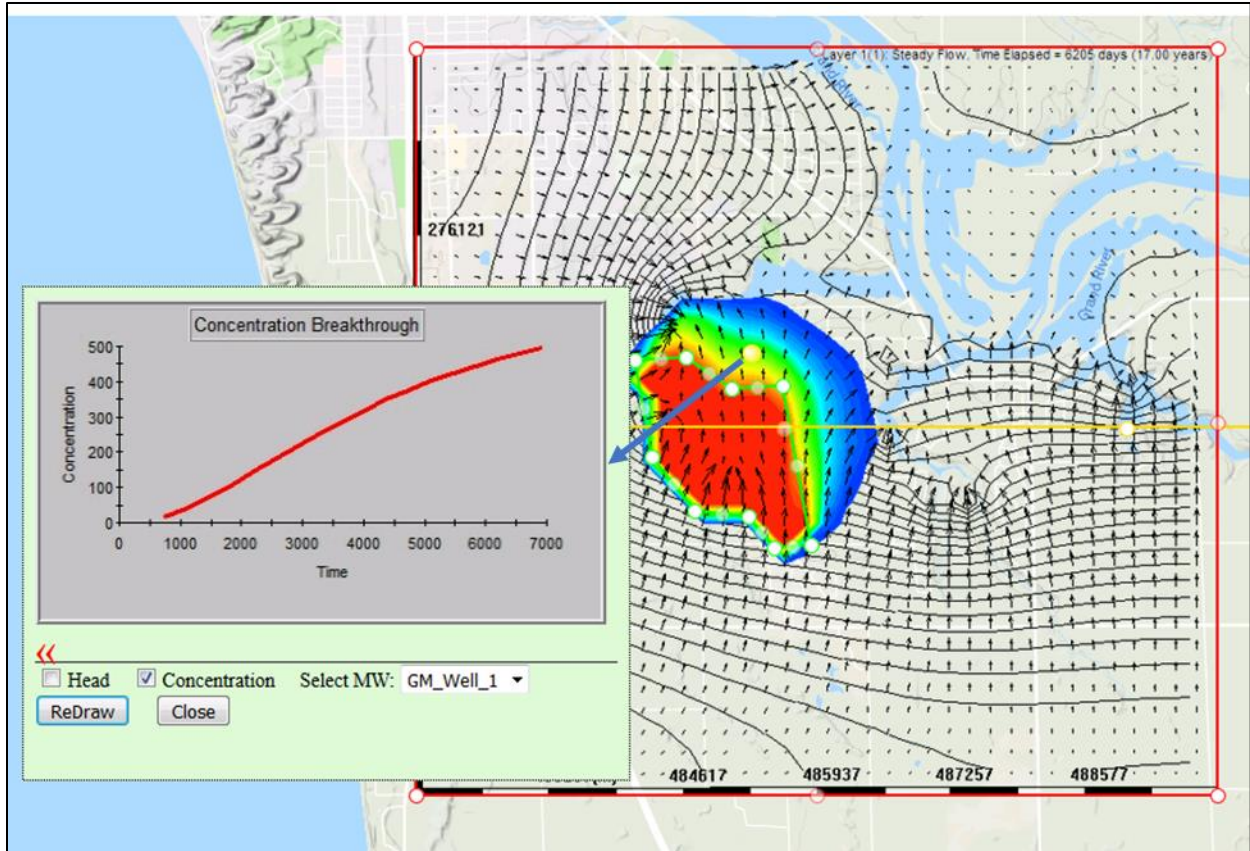


Figure 6-1: Example of using a monitoring well to generate a concentration breakthrough curve.

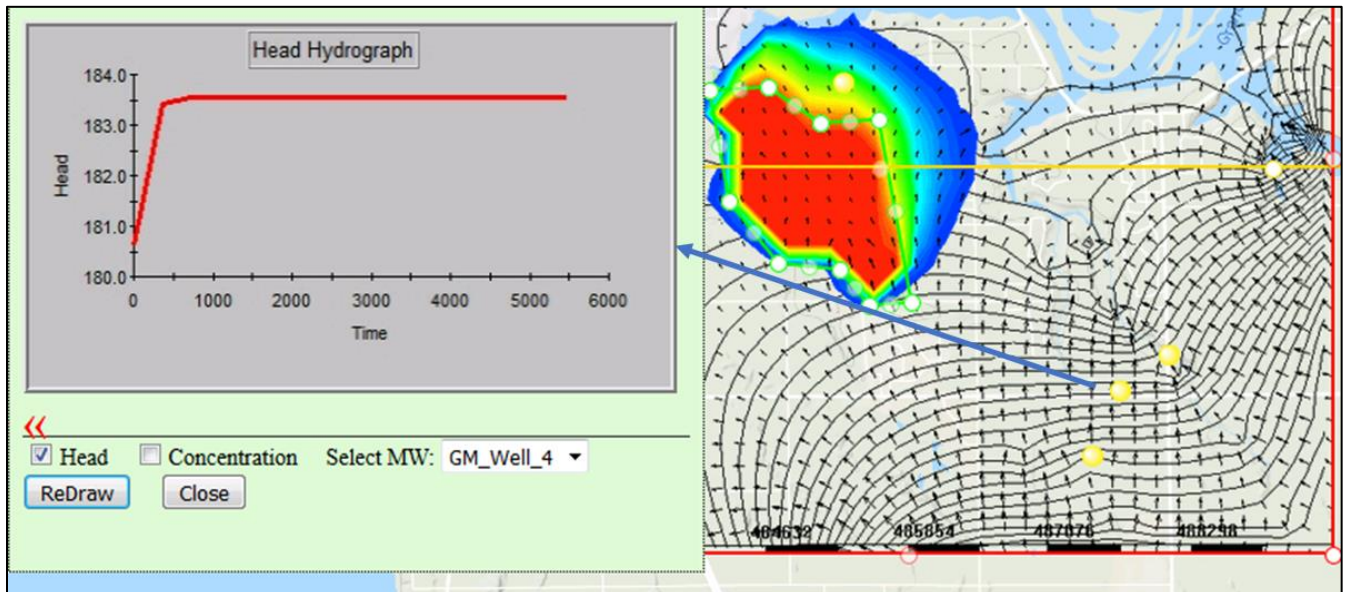


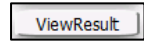
Figure 6-2: Example of using a monitoring well to visualize time-series simulated head data.

6.2 Water Balance Analyses

Zone budget (water balances) analyses can be conducted for:

- the entire model domain
- any zone added to the model domain
- a single grid cell of the model

A zone budget is automatically created for the model domain every time a model is submitted for simulation. As discussed in Section 5.4, to view Charts of model results, LM click on the ‘ViewResult’ button along the left-side of the MAGNET Modeling Environment, then select ‘Display Charts’ from the pop-up menu. This launches five different charts, including the Mass Balance Bar Chart.



Example Mass Balance Bar Charts are shown in Figure 6-3. Positive fluxes represent sources of water to the zone/domain, while negative fluxes represent sinks of water. Default units are cubic feet per second (CFS), but users can change the units to gallons per minute (GPM) or cubic meters per day (m³/day) by using the drop-down menu next to ‘Unit’. Any changes made require that the ‘ReDraw’ button is selected before the chart is updated.

Directly below the Mass Balance Bar Chart, the fluxes of the different budget terms are listed so that the user may export the information, if desired.

The following defines the different terms possibly present in a water balance and/or text output window:

- Bnd – water flux entering or leaving the zone through a boundary
- Csth – water flux from or moving to a constant head boundary condition
- Well (in bar chart) – water flux injected or removed from a injection/pumping well
- Rech – water flux added to the zone/domain from recharge
- Drain – water leaving the zone/domain through one-way head-dependent boundary conditions (drain or surface seepage)
- Error – residual of the zone budget. This is typically very small, unless there are issues with the model conceptualization or solution process.
- East – water flux added/removed through the eastern lateral zone boundary
- West – water flux added/removed through the western lateral zone boundary
- North – water flux added/removed through the northern lateral zone boundary
- South – water flux added/removed through the southern lateral zone boundary
- Top – water flux add/removed through the top cell/zone boundary (for multi-layer models)
- Bottom – water flux add/removed through the nbottom cell/zone boundary (for multi-layer models)
- Qwell (in text output window) - water flux injected or removed from a injection/pumping well
- Qrech (in text output window) - water flux added to the zone/domain from recharge
- Qriv
- Qdrn
- Qhde –
- Qevt – flux removed due to evapotranspiration
- Qret – flux added from return flow.

If the model domain lacks and zone conceptual features, the Mass Balance Bar Chart will show the zone budget for the entire model domain (see top-right graphic in Figure 6-3).

Select the radio button next to ‘Zone’ to show the zone budget for any zone(s) within the model domain. The drop-down menu allows for selecting from different zones in the model domain, if applicable (see

bottom-left graphic in Figure 6-3). See subsection 4.2.1 for details on making sure a zone budget is created for a zone conceptual feature.

Select the radio button next to 'Cell' to show the zone budget for an individual grid cell (see bottom-right graphic in Figure 6-3. Use the arrows next to 'I=', 'J=', and 'K=' to specify the model cell location of interest²¹

²¹ The origin for the I and J coordinates is the bottom-left vertex of the model domain. In MAGNET, '1' corresponds to the bottommost layer in a MAGNET model, while the largest number corresponds to the topmost layer in a MAGNET model.

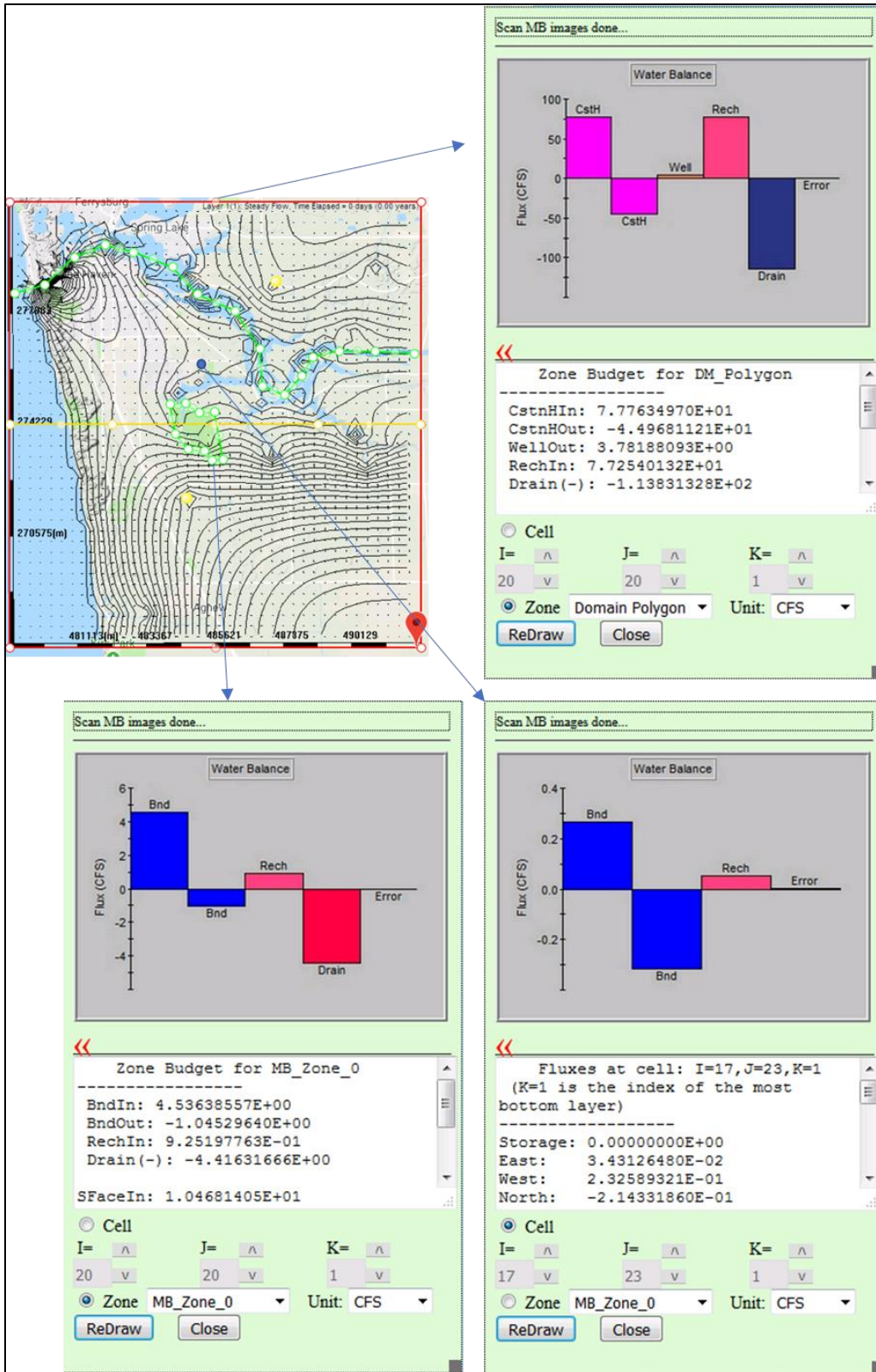
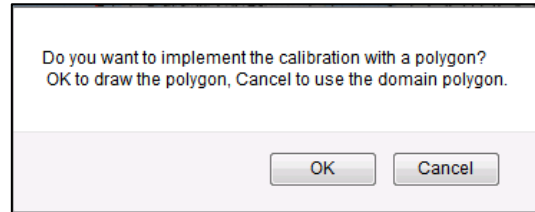
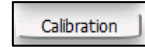


Figure 6-3: Example Zone (water) Balance charts. Top-right:: zone budget for entire domain polygon; bottom-left: zone budget for a zone added to the model domain; bottom-right: zone budget for a single model cell.

6.3 Model Results vs. Observations (Manual Calibration)

After model simulation is completed (in the case of steady state modeling) or paused (transient modeling), simulation head results (either for the steady-state flow field or the flow field of the current time step) can be compared to available observations by selecting the ‘Calibration’ button along the left-side of the MAGNET Modeling Environment (see Section 2.7). This will prompt the user to use a subregion of the model for comparison (choose ‘OK’ in the prompt) or the entire model domain (choose ‘Cancel’).



6.3.1 Loading Observational Data

Whether a subregion or the entire model domain is used for comparison, the 45 Degree Line Based Calibration Chart will appear after a selection is made (see Figure 6-4). The next step is to select the data source for observational data. Access the drop-down menu next to ‘Obs Data Source’ and select from one of the following options:

- Table – enter observational data directly into the interface
- File – import observational data from a properly formatted .txt file
- IGW Server – import observational data from the IGW server²²

²² In the current version of MAGNET, this option is only available for the State of Michigan.

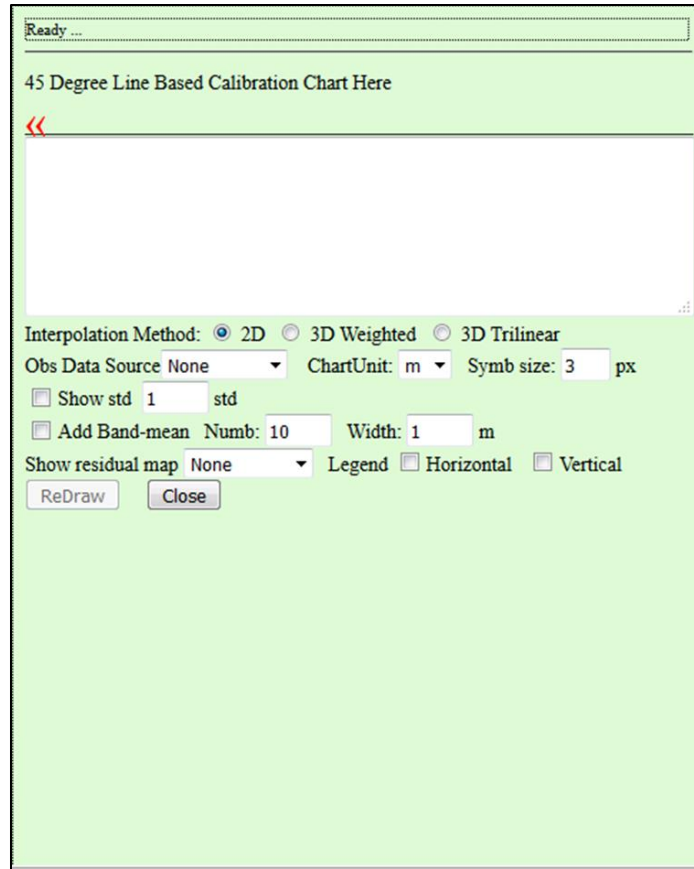


Figure 6-4: The 45 Degree Line Based Calibration Chart used for “manual calibration” (comparing current model outputs with observations).

Table Option. If the Table option is selected from the drop-down menu, an empty Calibration Data window will appear (see Figure 6-5). The first (header) line must include the following text:

WellID,Time,X,Y,Zf,Zt,LyrIndex,Obs,Color

One observation per line may be added after the first line.

- the 1st value is the well ID (WellID)
- the 2nd is ‘0’ (to indicate that the comparison is for steady state, or the current time step in the case of transient modeling (Time))
- the 3rd is the x-coordinate of the observation (X), e.g., longitude
- the 4th is the y-coordinate of the observation (Y), e.g., latitude
- the 5th is the well screen top elevation (Zf), in units of meters
- the 6th is the well screen bottom elevation (Zt), in units of meters
- the 7th is the layer index (LyrIndex), which can be used if the well is to be placed in a specific geological/computational layer in the case that well screen top and bottom elevation information is not available. By default, this value is set to ‘0’ so that the Zf and Zt values are used
- the 8th is the head observation/field measurement (obs), in units of meters
- the 9th is used to specify long-type RGB (Red-Green-Blue) color of the data markers in the calibration plot

An example of the correct header line and a line of observation inputs is shown in Figure 6-6. After adding inputs for all observations, use the ‘Click to Add Data’ button, then click ‘OK’ to use the data points in the calibration chart. Click on the ‘ReDraw’ button to plot simulation results versus the observation data (see next subsection).

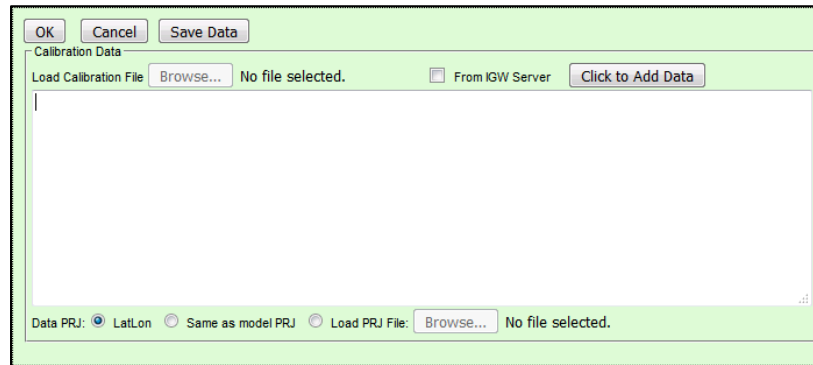


Figure 6-5: Calibration Data window, ready for direct input of observational data.

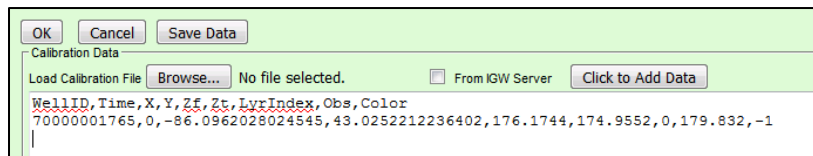


Figure 6-6: Example header line and line of observation inputs.

File Option. If a .txt file with the appropriate format is available on user’s local machine or network drive, it can be loaded into the Calibration Data window. Select File from the drop-down menu next to ‘Obs Data Source’. When the Calibration Data window appears, click the “Browse...” button to launch a file browser and navigate to and select the appropriate file for importing. An example Calibration Data window after importing observation data is shown in Figure 6-7.

To use the imported data for comparison to model results, click the ‘Click to Add Data’ button, then click ‘OK’ to use the data points in the calibration chart. Click on the ‘ReDraw’ button to plot simulation results versus the observation data.

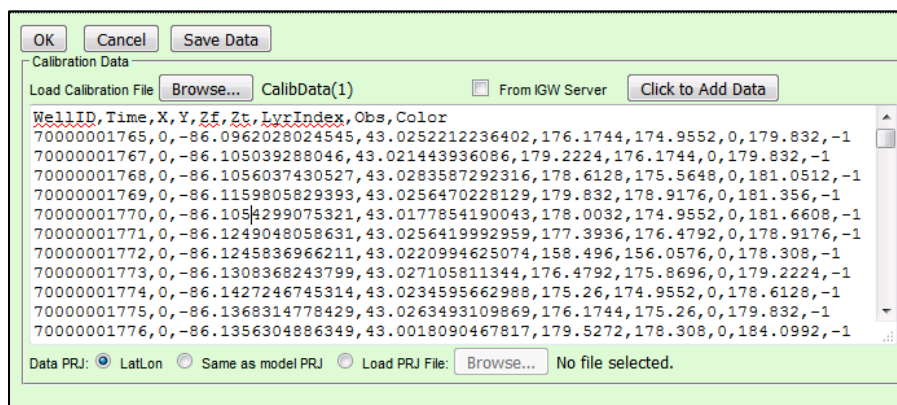


Figure 6-7: Calibration Data window after importing observation data from a file.

IGW Server Option. In the current version of MAGNET, water wells with measurements of groundwater levels are available for comparison with simulated head. If the IGW option is selected from the drop-down menu, the Filter for Well Data window will open (see Figure 6-8). This window allows the users to choose which types of wells to from which to import water level observations import:

- Drift(D) – include wells of any type completed in unconsolidated sediments associated with glacial drift and debris
- Bedrock(R) – include wells of any type terminated in consolidated materials/fractured rock
- Public Supply – water supply wells
- Industrial – wells supporting industrial or commercial activities
- Irrigation – wells used for irrigation needed for agriculture/landscaping
- Residential – domestic-use water wells
- Unknown(U) – water wells without use information
- Other – water wells used for some other purpose than those listed above

Check the radio box next to each option to include it in the well selection for importation, or uncheck to filter it out from the import.

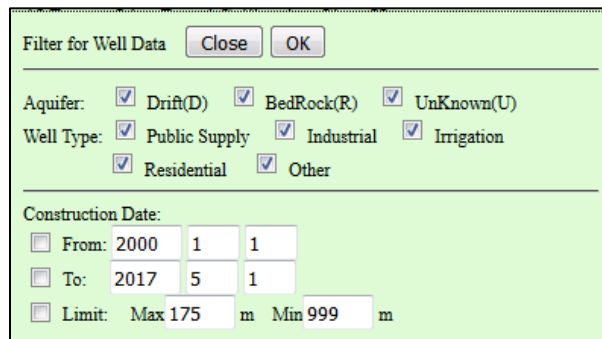
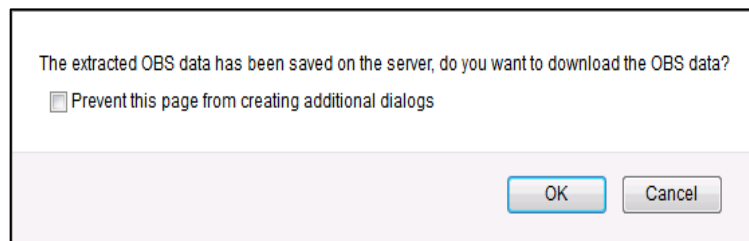


Figure 6-8: Filter for Well Data window for importing data from the IGW server

The user may further refine the well selection by specifying a time interval from which to select wells on the basis of well construction date. To include a filter for the earliest date of construction, check the radio box next to ‘From:’ and enter the starting date (year, month, day) in the text boxes provided. To include a filter for the latest date, check the radio box next to ‘To:’ and enter the ending date (year, month, day) in the text boxes provided.

Lastly, the user may restrict the well selection by a specified data range. Check the radio box next to ‘Limit:’, then enter the maximum and/or minimum head values for import into the text box next to ‘Max’ and/or Min (units: meters), respectively.

After all filters have been selected/applied, click ‘OK’ to continue importing data from the IGW server. A prompt will appear asking if data should be downloaded for further analysis (select ‘OK’) or if it should just be used for display in the Calibration chart (select ‘Cancel’).



Click on the ‘ReDraw’ button to plot simulation results versus the observation data.

After the observational data has been imported and 'ReDraw' has been selected, the Modeling Samples vs. Calibration plot and the available chart options will appear. Additionally, any downloaded data will be displayed in the text window below the plot, which can be utilized for offline analysis if desired (see Figure 6-9).

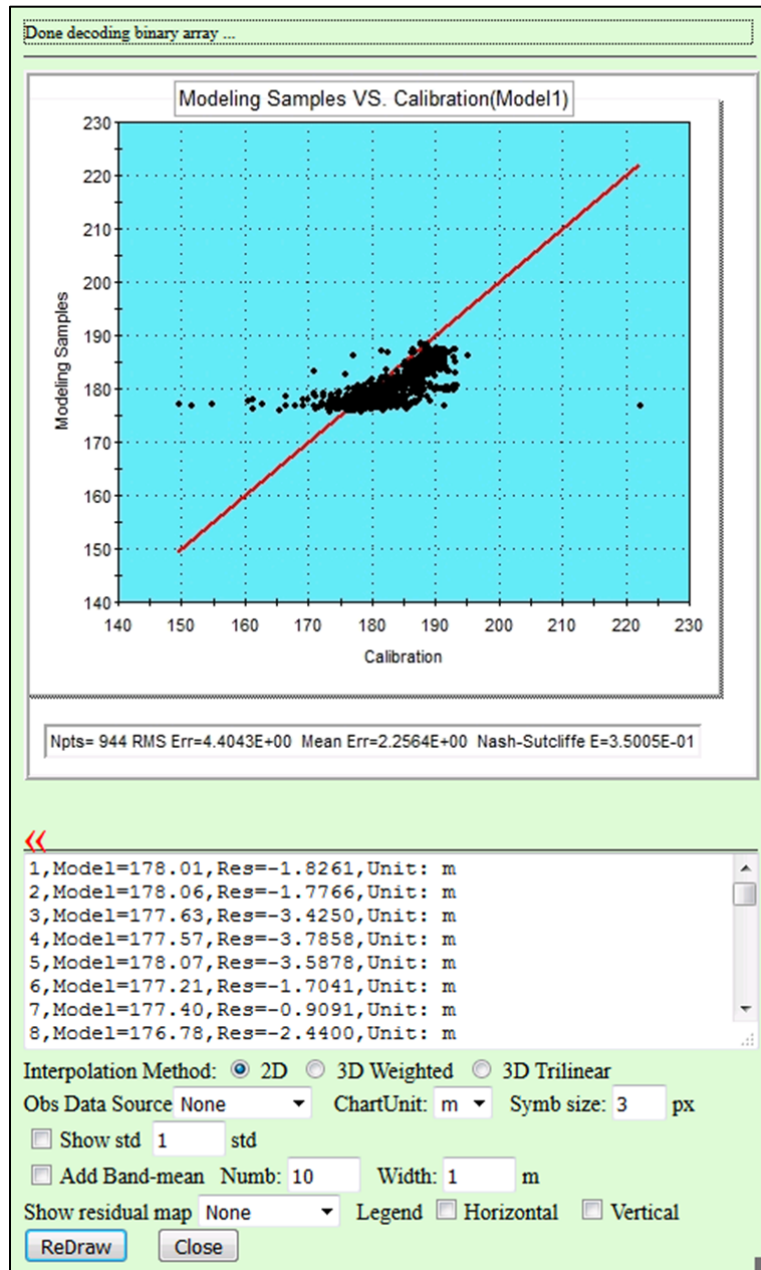


Figure 6-9: Modeling Samples vs. Calibration plot, extracted model values and residuals (relative to the observations), and chart options.

Each data point provides information on the agreement between the simulated head and the observed head. Points that fall directly on the 45 degree line represent perfect agreement between the model and measurement. Points above indicate a modeling result that is larger than the observed value, while points below indicate a modeling result that is smaller than the observed value.

Statistical indicators are provided directly below the plot:

- Npts – the number of observational data point available for the comparison
- RMS Err – the root-mean-square error (units: meters)
- Mean Err – the mean error (units: meters)
- Nash-Sutcliffe E – the Nash-Sutcliffe coefficient ($0 \leq E \leq 1$)

Interpolation Methods. Because water level measurements in wells never/rarely occur at the location of an model node, interpolation of simulated head values is needed to compare to observations. Three options are available in MAGNET:

- 2D (traditional) bi-linear interpolation
- 3D Trilinear interpolation
- 3D Weighted interpolation for multi-layer models, where the weighting is determined by screen-transmissivity (see Figure 6-10).

By default, 2D interpolation is used.

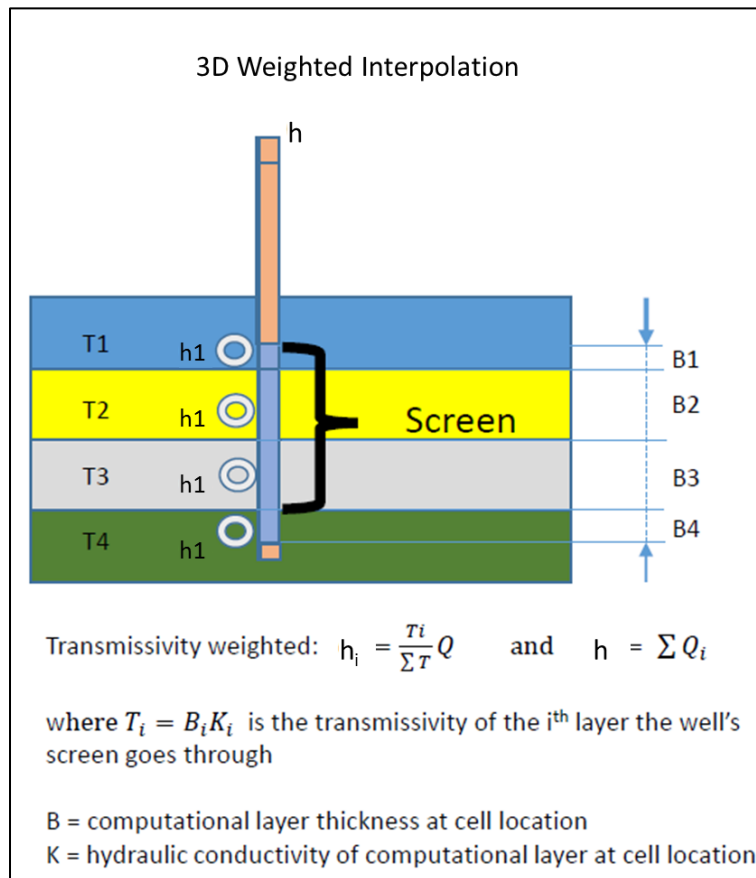


Figure 6-10: Scheme used for 3D Weighted interpolation of simulated head for comparison with observations.

Chart Options. The user may adjust the axes units, the data marker size, and add statistical indicator overlays directly to the Calibration plot.

To change the axes units between meters and feet, use the drop-down menu next to 'ChartUnit:'. To change the size of the data markers (units: display pixels), click in the text box next to 'Symb size' and enter the desired value.

To add confidence intervals to the Calibration Plot, check the radio box next to 'Show std' and enter the number of standard deviations used for computing the confidence interval in the text box provided. Additionally, users may add a moving-window, "Band-mean" average to the Calibration plot by checking the radio box next to 'Add Band-mean'. The value entered in the text box next to 'Numb:' is the number of equal-sized "bands" to divide the data range into for computing the average simulated and observed head within each band. Alternatively, users may input the size of the band by entering a value in the text box next to 'Width'²³. Figure 6-11 provides an example of adding Confidence Intervals and the Band-mean markers.

²³ If a non-zero value is entered in the 'Numb' box, this option will override the 'Width' option.

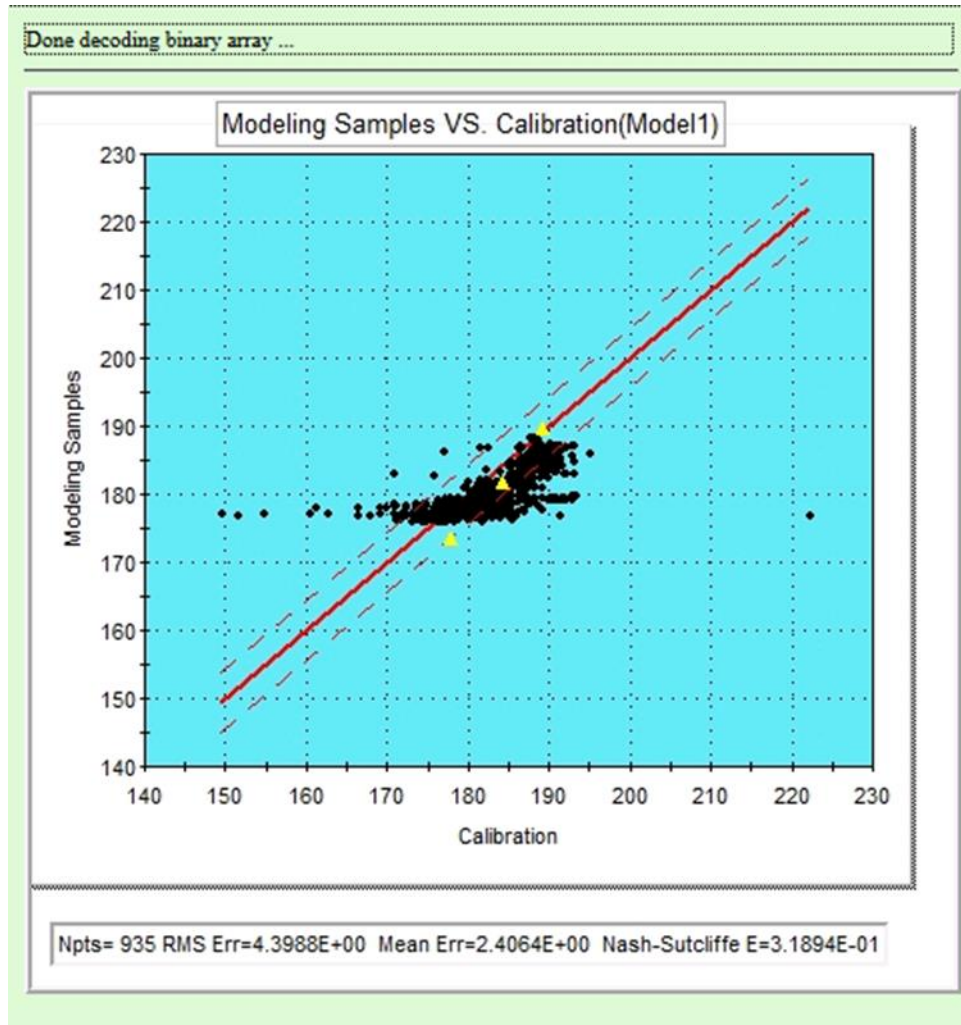


Figure 6-11: Example Calibration plot with Confidence intervals (red dotted line) of one standard deviation and Band-mean indicators (yellow triangles).

Residual Maps. In MAGNET, it is possible to display the locations of comparison points which yield residuals above/below a prescribed threshold. Use the drop-down menu next to ‘Show residual map’ to apply one of the following mapping options:

- Distribution – the residual for all comparison points is indicated by colored data markers
- Outside std – the location of comparison points that yielded residuals larger than the number of standard deviations used for computing confidence intervals CI).
- Inside std – the location of comparison points that yielded residuals larger than the number of standard deviations used for computing confidence intervals.

Each of the possible residual mapping options are shown in Figure 6-12. If the Distribution option is chosen, it is useful to check the radio box next to ‘Horizontal’ or ‘Vertical’ to display a legend in the horizontal or vertical direction, respectively. When the ‘Outside std’ option is selected, comparison points yielding residuals that are outside the CI and with a simulated head less than observed head are displayed as blue triangles in the MDWA. Conversely, comparison points yielding residuals that are outside the CI and with a simulated head greater than observed head are displayed as red triangles (see Figure 6-12).

When the 'Inside std' option is selected, comparison points yielding residuals that are inside the CI and with a simulated head less than observed head are displayed as blue triangles in the MDWA. Conversely, comparison points yielding residuals that are inside the CI and with a simulated head greater than observed head are displayed as red triangles (again, see Figure 6-12).

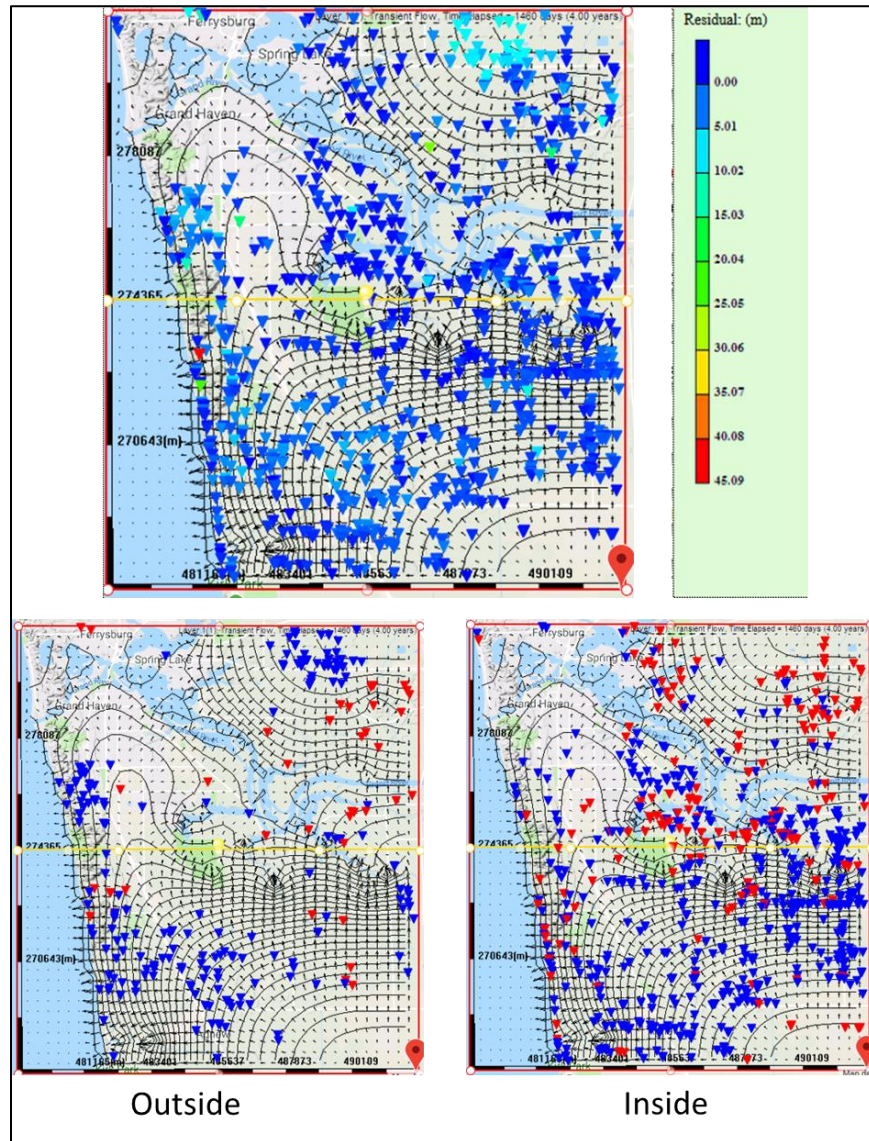


Figure 6-12: Examples of residual mapping display options.

Chapter 7 UTILITIES & FILE FUNCTIONS

This chapter describes tools/features used for saving or loading models, publishing and editing models and other file functions available in MAGNET. These capabilities are accessed in the Utilities sub-menu (Figure 7-1).

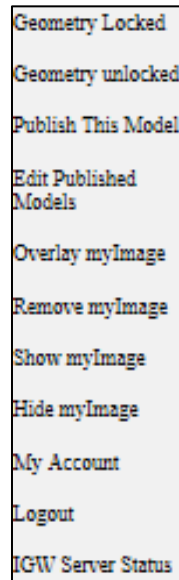
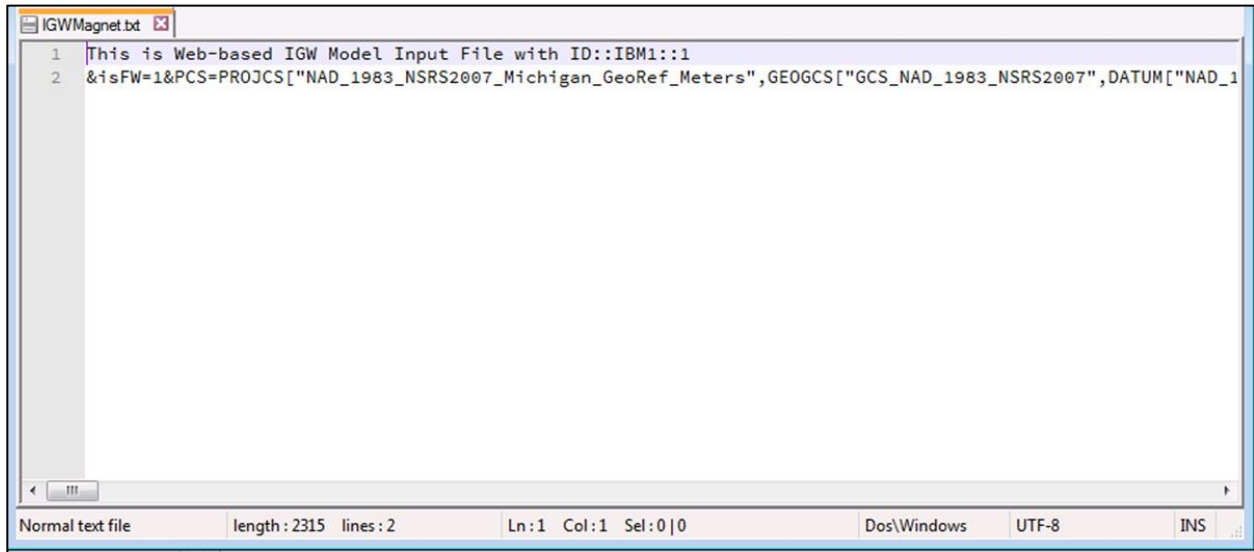
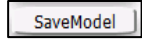


Figure 7-1: Utilities sub-menu.

7.1 Saving and Loading Models

In the current version of MAGNET, the conceptual model (i.e., the model domain and all of the conceptual features and parametrization options) can be saved for simulation/analysis/refinement at a later time. This can be done before submitting the model for simulation, after the simulation is completed (steady flow modeling) or when the simulation is paused (unsteady modeling).

To save a conceptual model, click the ‘SaveModel’ button along the left-side of the MAGNET modeling environment. This will launch the ‘Opening IGWMagnet Window) to

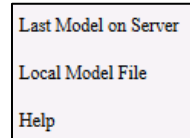


save the MAGNET model file (ASCII/.txt format) to the local machine or directly open it with a ASCII/.txt editor. An example MAGNET model file is shown in Figure 7-2.²⁴

Figure 7-2: Example MAGNET model file.

To load a MAGNET model file, click the ‘LoadModel’ button along the left-side of the MAGNET modeling environment. This will open a sub-menu with three options:

- Last Model on Server – which allows the latest conceptual model saved with MAGNET user account to be loaded;
- Local Model File – which allows the user to browse to and load a MAGNET file saved on their local machine;
- Help – this feature is not available in the current version of MAGNET.



7.2 Clear Browser Cache

As with other web-based applications, it may be necessary to clean your browser history data between MAGNET modeling sessions, specifically, your cached data and files. This process is different for different web-browsers, but is typically a straightforward process. For example, to clear cache data and files in Microsoft Edge (for Windows 10), go to Settings, click ‘Choose what to clear’, then make sure ‘Cached data and files’ is checked and click ‘Clear’.

7.3 Publishing & Editing MAGNET Network Models

A powerful capability of MAGNET is to allow users to publish their model, results, and data on the MAGNET network to be viewed by other users. Specifically, users can publish/share: MAGNET model files (see previous section); image files of model results (e.g., head distributions, cross-sections, monitoring

²⁴ In future versions of MAGNET, user will be able to upload and share their models, data, and tools if desired.

well plots, water balances); and/or animation files²⁵ (e.g., transient head dynamics, particle tracking or contaminant transport). Users can decide whether to publish the conceptual model or just results (images or videos) and/or comments related to the model. If the conceptual model is published, other users can directly load it for editing on the MAGNET modeling web interface.

After creating and simulating a model, click the ‘Utilities’ button and then select ‘Publish This Model’ to launch the Model Publishing Options window (see Figure 7-3). This window allows uploading three image files related to model results and one video animation. Users should also provide a Model Name and a Model Description in the text boxes provided.



To only share results, but not the conceptual model, check the box next to ‘Do not publish conceptual model file’. In fact, if users simply want to display the results of a model created with some other software/tool, they can simply draw a box presenting the modeling area, and proceed right to the Model Publishing Options window without submitting the model for simulation. The centroid location of the model (latitude and longitude, or Clat, Clong, respectively) will be used for further editing (see subsection 7.3.1).

After adding all text/files, click the ‘Publish’ button to add the model and/or model results to the MAGNET Model Network. Or click the red ‘X’ to cancel the publication process.

²⁵ Images and videos to be uploaded need to be prepared by the user, not the platform, either through screen capturing or using a commercial movie maker.

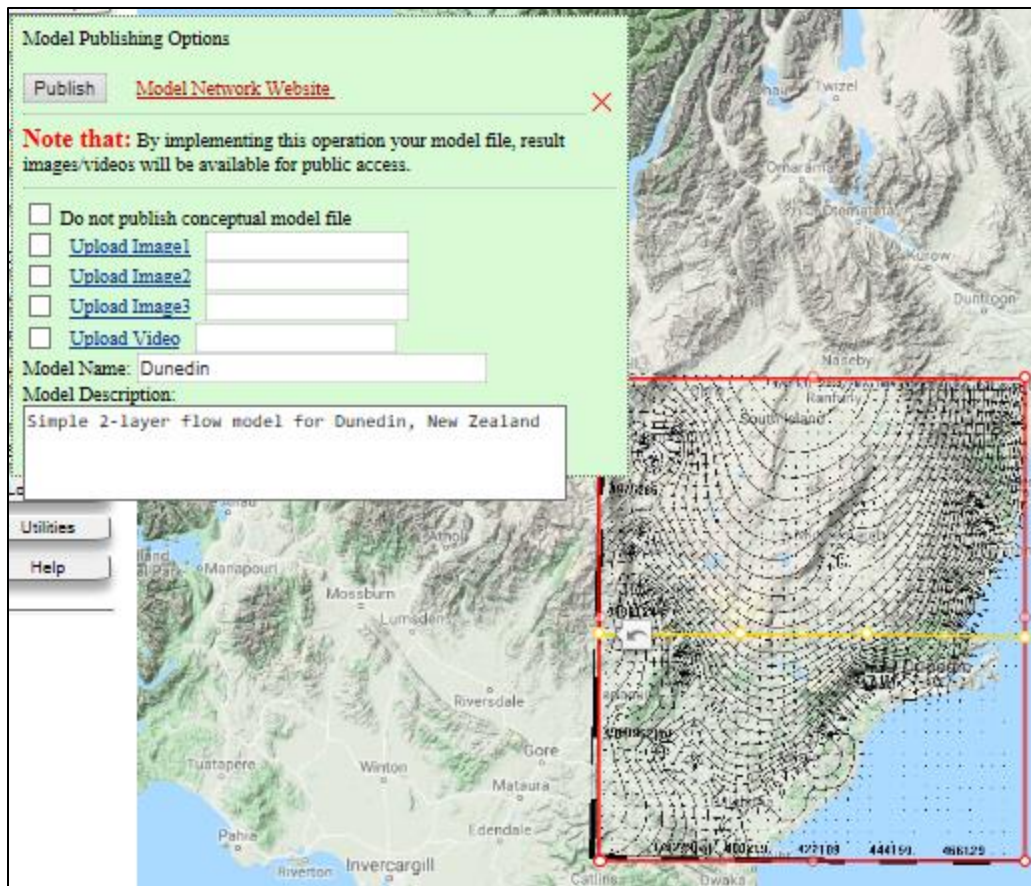


Figure 7-3: Example of publishing a model to the MAGNET network using the Model Publishing Options window.

7.3.2 *MAGNET Modeling Network*

The MAGNET Model Network (Figure 7-4) can be accessed by clicking the ‘Model Network Website’ link in the Model Publishing Options window, or by navigating to:

<http://www.magnet4water.com/modelnetwork/>

The different models created around the globe are shown as blue pin markers in the map display of the MAGNET Model Network. Comments about the platform/model network can be submitted to Hydrosimulatics using the interface in the bottom-left. Posted comments on are shown in the bottom-right.

The screenshot displays the Magnet Model Network interface. At the top, the browser address bar shows the URL www.magnet4water.com/modelnetwork/. The main heading is "Model Location Maps (Number of models=232)". Below this is a Google Map of the world with numerous blue pins indicating model locations across various continents. To the right of the map is a "Model Description" panel, which currently shows "Not available". Below the map is a search bar with the text "Click a marker to view model results and comments. Or search a model name here" and a "Search" button. A footer note states "Developed by HydroSimulAtics, INC. 2018 Last Updated: 08/15/2018". At the bottom, there is a "Comments on our platform" section with a form for "Your Name:" and a "Submit Comment" button, and a list of user comments with their dates and ratings (all 5 stars).

Figure 7-4: MAGNET Model Network, available at www.magnet4water.com/modelnetwork/.

Zoom to and LM click on any pin to view more details about the model (date created, modeler/username, and latitude/longitude of the centroid of the model domain), and to access the model files/results (see Figure 7-8). Alternatively, users can use the search box to search for model name.

Click the 'Model File' link to download the conceptual model file to a local drive so that it can be at loaded at a later time for further development/analysis (see Section 7.1).

Click the 'Load Model to IGW Magnet' to directly load the model to the MAGNET modeling environment (in a new web page/tab).

Click the ‘Model Image’ or ‘Model Video’ to open the model image or video file(s) in a new web page/tab.

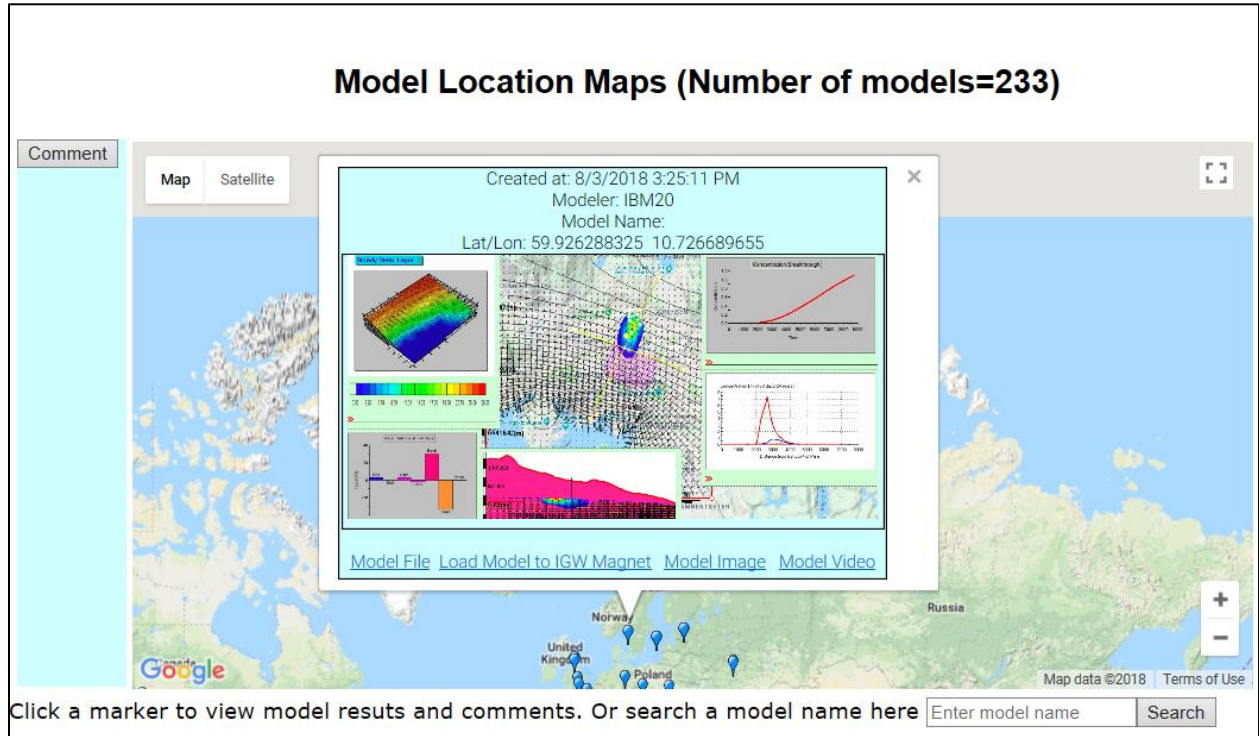


Figure 7-5: Selected model details in the MAGNET Model Network web interface.

If the user clicks in the center part of model details window, the model results and comments page will launch (see Figure 7-6). This page allows for viewing the uploaded images and/or videos in greater detail. Visitors may also comment on and “rank” the model by visiting the comment interface in the bottom-left portion of the web-page. Comments will be sent directly to the model creator via email, and will be displayed in the right-hand-pane of the comment interface.

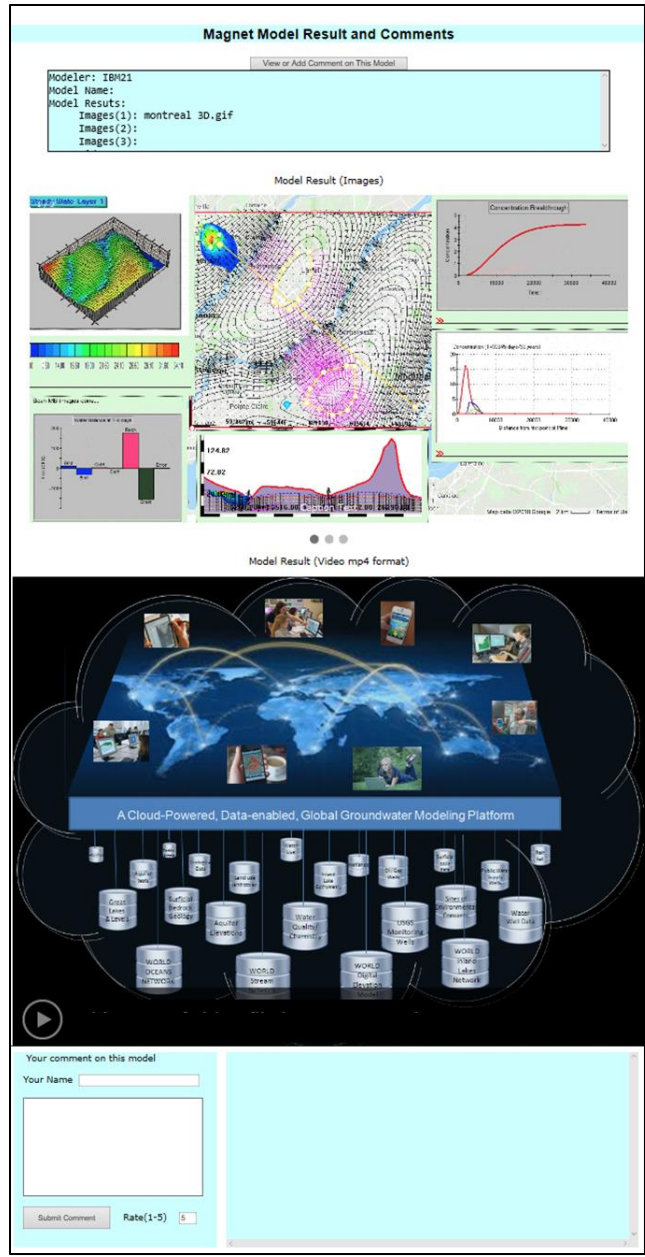


Figure 7-6: Model Results and Comments page.

7.3.3 Editing Your Published Models

At any time, users can edit the settings of any of their published models by clicking the ‘Edit Published Models’ button in the Utilities submenu²⁶. This opens the Edit Published Model Options window (Figure 7-7).

Edit Published Models

²⁶ This requires that MAGNET users login/sign-up to the MAGNET Modeling Platform (see Section 1.3).

First, users to select from their list of published models from the drop-down menu. Then they can:

- Replace the conceptual model file with the current model file (i.e., the model for the last job submitted)
- Update whether or not the conceptual model file is published
- Update/upload images or videos of the model
- Add/update model name or a model description.

Click 'Update This Record' to save the changes made, or click the red 'X' to cancel the editing process.

Clicking the 'Delete This Record' button will remove the selected published model from the MAGET Model Network.

Edit Published Model Options

Update This Record Delete This Record [Model Network Website](#)

Note that: By implementing the DELETE operation your model file, result images/videos will be removed from the server.

Your current records of published model(2):
None

Replace with current model file
CLat= CLon=
Created DateTime:

Do not publish conceptual model file
 Upload Image1
 Upload Image2
 Upload Image3
 Upload Video

Model Name:
Model Description:

Figure 7-7: Window for editing user published models.

7.3 Background Images

In some cases, it may be helpful/necessary to display an image layer in the MDWA (e.g., site drawings, GIS layers/maps of land or subsurface features, etc.). Use the ‘Utilities’ button along the left-side of the MAGNET modeling environment to access the sub-menu options for background images.



Begin by using ‘Overlay myImage’ option to open the ‘Overlay an Image on Maps’ menu (see Figure 7-8). The text following “Note that:” provides details on the image requirements and the directions for specifying the image pathname if the web browser does not support local resource accessing. If the browser does support local resource accessing, click on the ‘Browse...’ button to navigate to the image location and open the image. Note that when using the ‘UpLoad’ option, the user must specify the overlay extent using the text boxes next to ‘Top Latitude’, ‘Right Longitude’, etc.

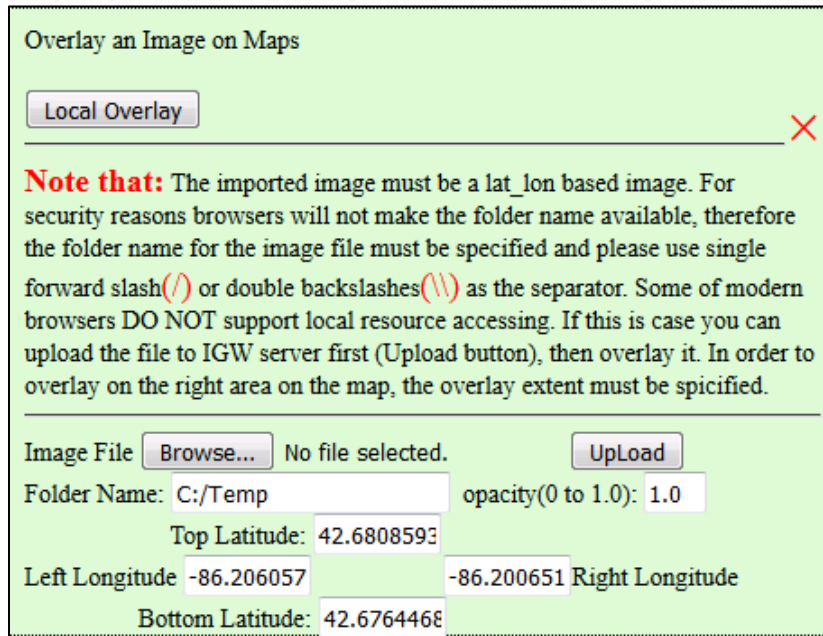


Figure 7-8: 'Overlay an Image on Maps' menu.

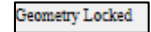
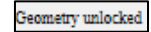
Similarly to hiding and re-displaying simulation results (see subsection 5.2.4), background images can be removed and re-displayed in the MDWA. Use the ‘Hide myImage’ option and the ‘Show myImage’ option from the Utilities sub-menu to hide and re-display the background images, respectively. To completely remove the background image from the MDWA, use the ‘Remove my Image’ option from the Utilities sub-menu.

The ‘Down/UpLoad File button is disabled in the current version of MAGNET.

7.4 Geometry Lock/Unlock

When a user draws a conceptual feature (polygon/zone, polyline, etc.), nodes (small circles are automatically displayed along the line segment/perimeter for “instant” editing of the geometry by click-dragging nodes into desired locations. However, in some cases displaying these nodes will obstruct a key conceptual feature or part of the background image, and the nodes may be unintentionally moved if the nodes are accidentally selected by the cursor.

To disable this default setting, click the ‘Geometry Locked’ button. Doing so will “lock” all present geometric features (i.e., they are no longer editable or clickable) and feature nodes (circles) will no longer be displayed. Any new geometric features added will still be editable until the user clicks ‘Geometry Locked’ once more.

A rectangular button with a thin black border and the text "Geometry Locked" in a sans-serif font.A rectangular button with a thin black border and the text "Geometry unlocked" in a sans-serif font.

To make all geometric feature editable/clickable, click ‘Geometry unlocked’ from the Utilities menu.