



---

## MAGNET TUTORIAL

### Massive Plume Migration in Northern Michigan Community

This tutorial provides step-by-step instructions for simulating one of the largest groundwater contamination plumes in the United States – and the largest TCE (trichloroethylene) plume in Michigan. The source of the contamination is degreasing solvents used at the Wickes Manufacturing plant in the small town of Mancelona during the 1940s and '50s. Originally disposed into shallow pits on site, the TCE plume has migrated northwest in the surficial glacial aquifer, toward a well field providing drinking water to private properties and municipalities. TCE is a known carcinogen.

The following steps outline how use MAGNET to develop, calibrate, and apply a steady groundwater flow and transport simulator for predicting the migration of the TCE plume and simulating its removal with a groundwater purge well. Annotated graphics supplement concise written instructions. Users are encouraged to build their own model as they review the tutorial. Real-time help pages are available for the various tools/options used in the MAGNET modeling environment (see the '?' buttons or the 'Help' sub options available throughout the menus and interfaces).

NOTE: this tutorial assumes that the reader has created a free MAGNET user account. If this has not been done, see the MAGNET Quick Helper menu in the MAGNET modeling environment ('Help' > 'Quick Helper').

#### 1: Create model domains (regional and local) and load/overlay the site map

Simulating the plume transport and remediation operations requires detailed information about the groundwater flow at the site. But the site-specific flow conditions are influenced by the regional flow patterns, requiring some understanding of how groundwater head is distributed in the area around (and especially upstream) of the site. The proper way to handle this 'multiscale' nature of groundwater systems is to simulate large-scale conditions with a larger, relatively coarse-grid model – the Regional model. This model can then be used to inform a smaller, finer grid model (the Local model).

- Use a web browser to visit the MAGNET Groundwater Modeling Environment: <https://www.magnet4water.com/magnet> (Figure 1)
- Use the navigation tools to increase the zoom level to '10' and search for Mancelona, Michigan, United States (Figure 1). Hit Enter to zoom to the general area of the site.
- Use the 'DomainRect' tool ('Conceptual Model' > DrawDomain' > 'DomainRect') to draw a rectangular model domain of the site and surrounding region. This will be the Regional Model domain (see Figure 2).
- Use the 'ZoneRect' tool ('Conceptual Model' > 'Zones' > 'ZoneRect') to add the Local Model area, including the town of Mancelona and the area to the northwest. Click 'SaveShape' to finalize the shape and launch the Zone Attributes menu (Figure 3).

- In the 'Flow Property' tab Zone Attributes menu, check the radio box next to 'Submodel Domain' and make sure the 'PolygonOnly' option is selected. This will be changed to 'Active' once the zone is needed to model the local area at and around the site (after the regional model is calibrated – see below).
- Use the 'Overlay myImage' tool to add a site map of the delineated plume to the map display ('Other Tools' > 'Utilities' > 'Overlay myImage')
- Follow the instructions in the Overlay image menu (Figure 4).
  - Download and browse to the appropriate file: 'PlumeImage\_georef1'
  - Enter the image extents by copying coordinates from the following file: 'PlumeImage\_georef1\_extent.txt'
  - Upload the image and coordinates to your MAGNET user account.
  - Overlay the image to the map display.

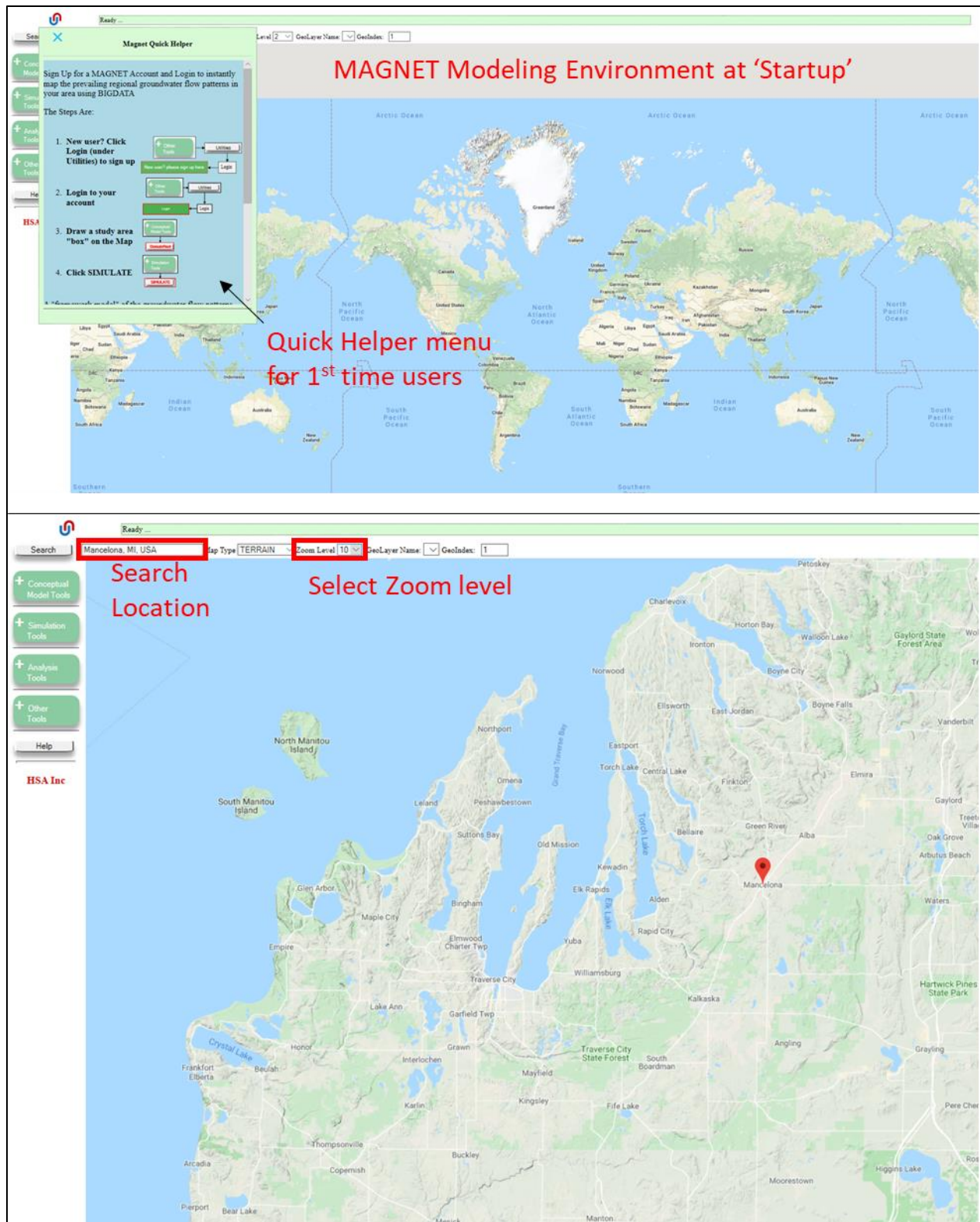


Figure 1. (Top) The default screen setting when first accessing the MAGNET Modeling Environment. First time users will need to create a *free* MAGNET user account (see the Quick Helper menu). (Bottom) Navigation tools to zoom to

the site of interest. After adjusting the zoom level and entering the site name in the Search bar, the user must hit 'Enter' to navigate to the site.

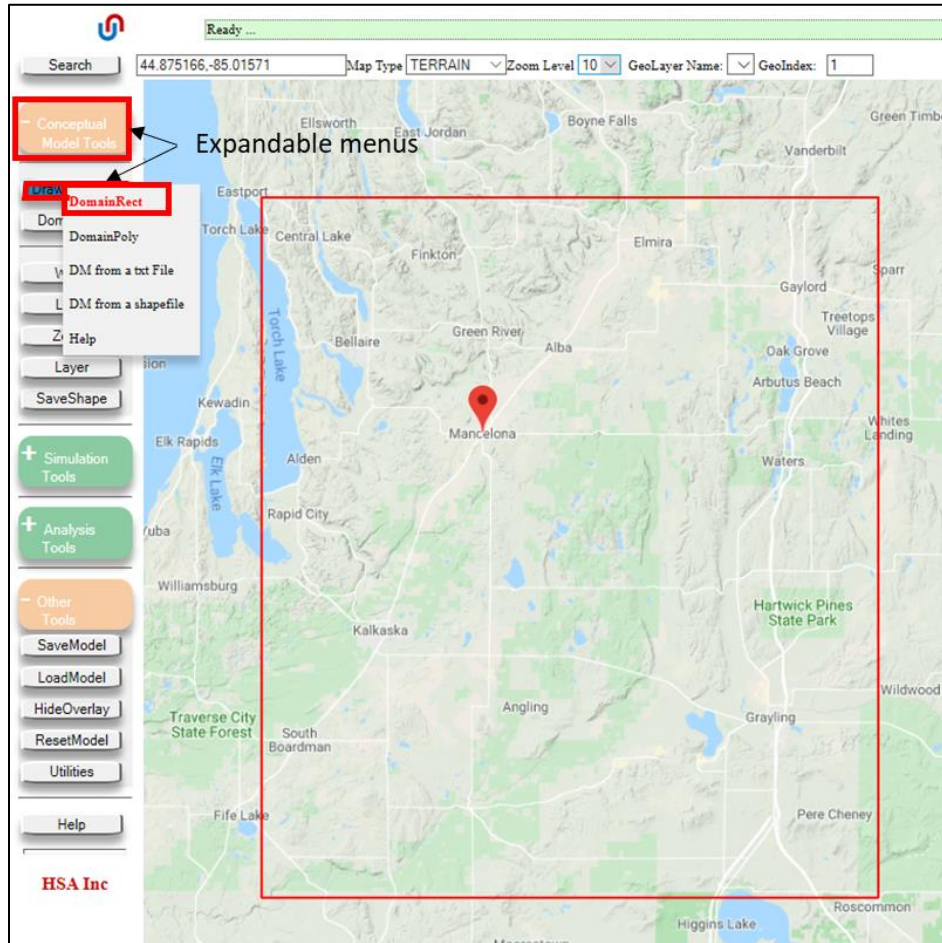


Figure 2. Drawing the Regional Model domain. When using the 'DomainRect' button, indicate the position of the first vertex with a single left-mouse (LM) click, and indicate the position of the vertex diagonal from the initial vertex with another single LM click. (Note: When drawing a polygon using 'DomainPoly', single-click the LM button to define the vertices of the polygon that will define the model domain.)



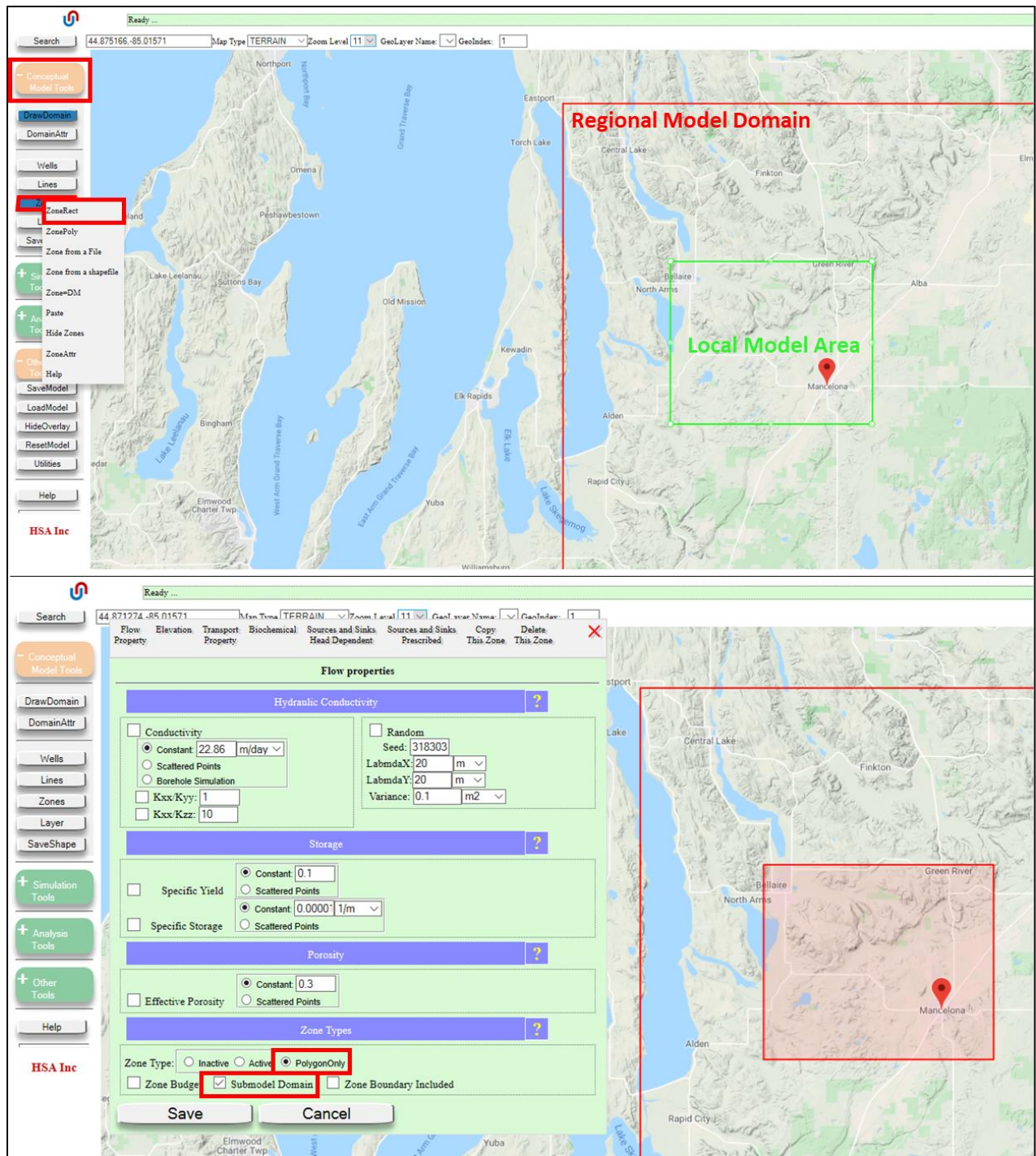


Figure 3. (Top) Adding a zone to represent the submodel, or Local Model. When using the 'ZoneRect' button, indicate the position of the first vertex with a single left-mouse (LM) click, and indicate the position of the vertex diagonal from the initial vertex with another single LM click. (Note: When drawing a polygon using 'ZonePoly', single-click the LM button to define the vertices of the polygon that will define the model domain. (Bottom) Using the Zone Attributes menu to assign the added zone feature as the submodel domain. Note that 'PolygonOnly' is selected to allow the Regional Model to be executed (i.e., the local zone is ignored). After running the Regional Model, 'Active' is selected to run the submodel the next time the job is submitted, with boundary conditions derived from the Regional model.

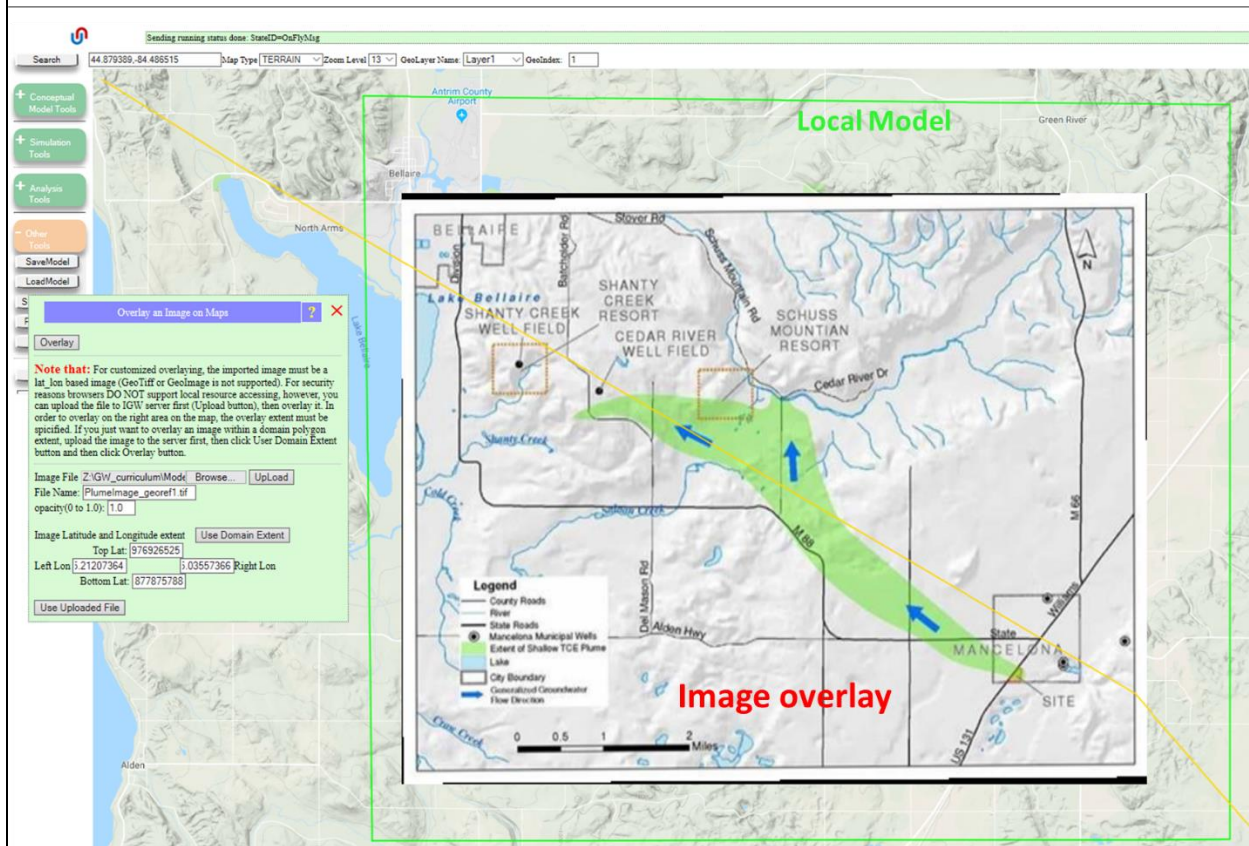
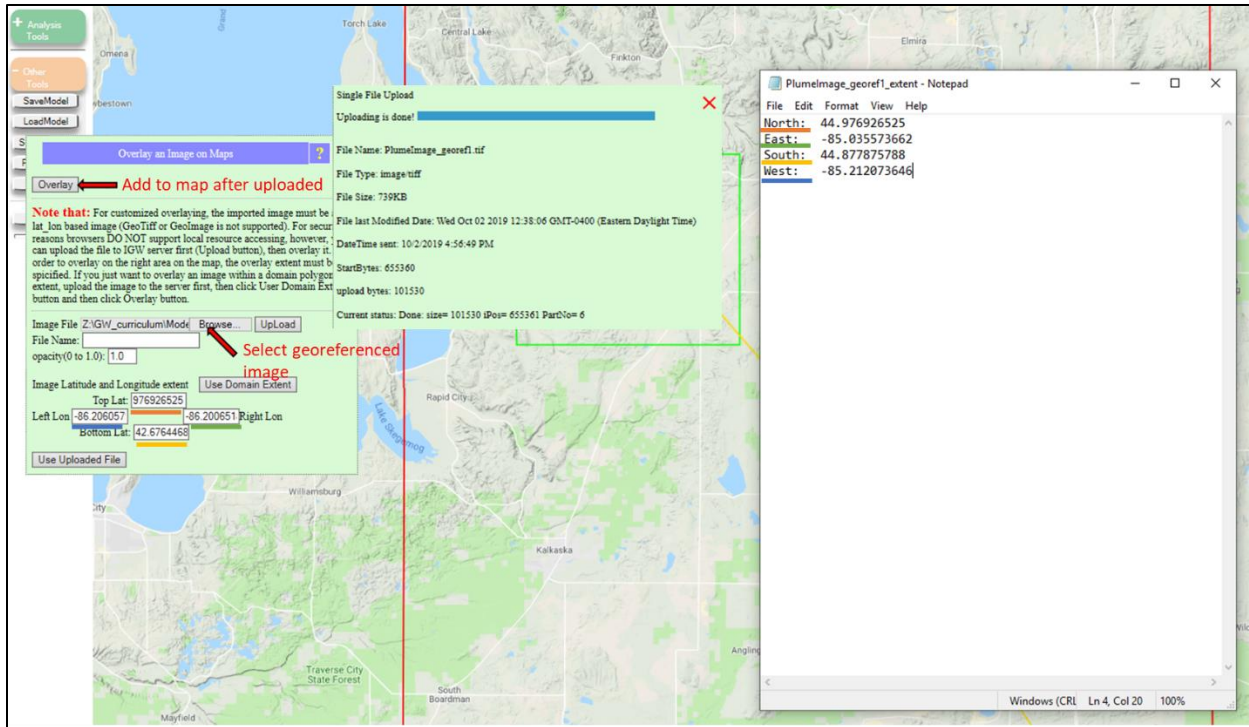


Figure 4. (Top) The 'Overlay an Image on Maps' menu and the steps needed to complete a site image overlay. First, select the georeferenced image from the local drive. Then input the image extent using the information from the

associated .txt file. Then select 'Upload' then 'Overlay' to add the image to the map display. (Bottom) The georeferenced site map added to the map display.

## **2: review/assign parameters of Regional model**

- The model inputs/parameters are assigned using the DomainAttr (Domain Attributes) tool. This menu also provides options for numerical simulation (e.g. grid size, time-step, etc.), display settings, and projection system, among other options. (Conceptual Model Tools > 'Domain Attr')
- Assign the aquifer elevations – the bottom and top surfaces – by checking the appropriate radio boxes.
  - The top surface should follow the land surface characterized by the 10m Digital Elevation Model stored on the Server.
  - The bottom surface should follow the bedrock surface underlying the surficial glacial deposits. This is available as a raster layer stored on the Server.
- The hydraulic conductivity should be input as a spatially-explicit data layer stored on the Server ('DriftK2' for the State of Michigan).
- The long-term, average recharge should be assigned as a spatially-explicit data layer stored on the Server (for the State of Michigan).



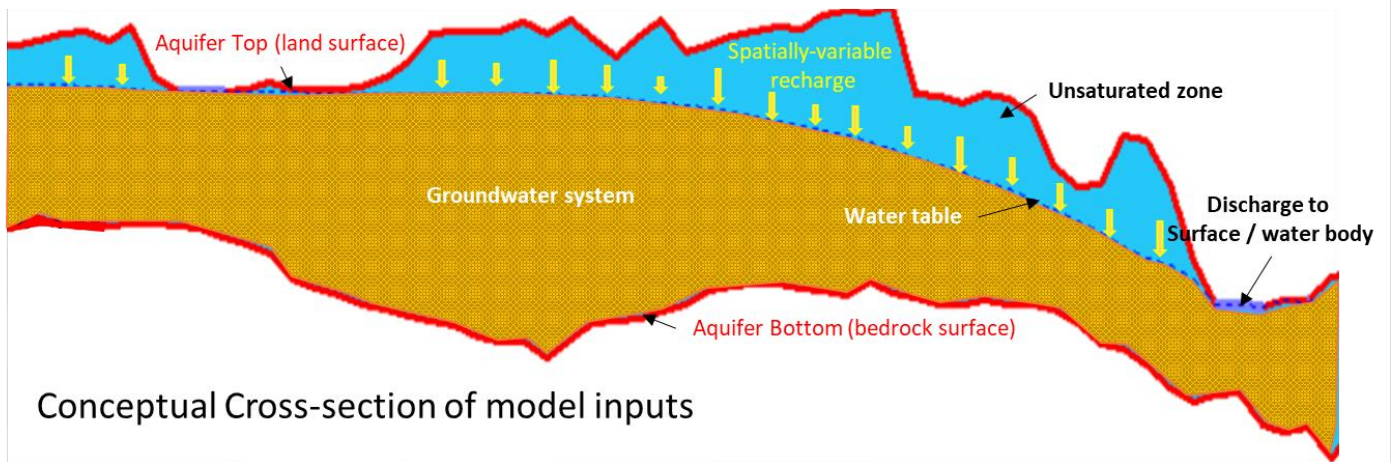
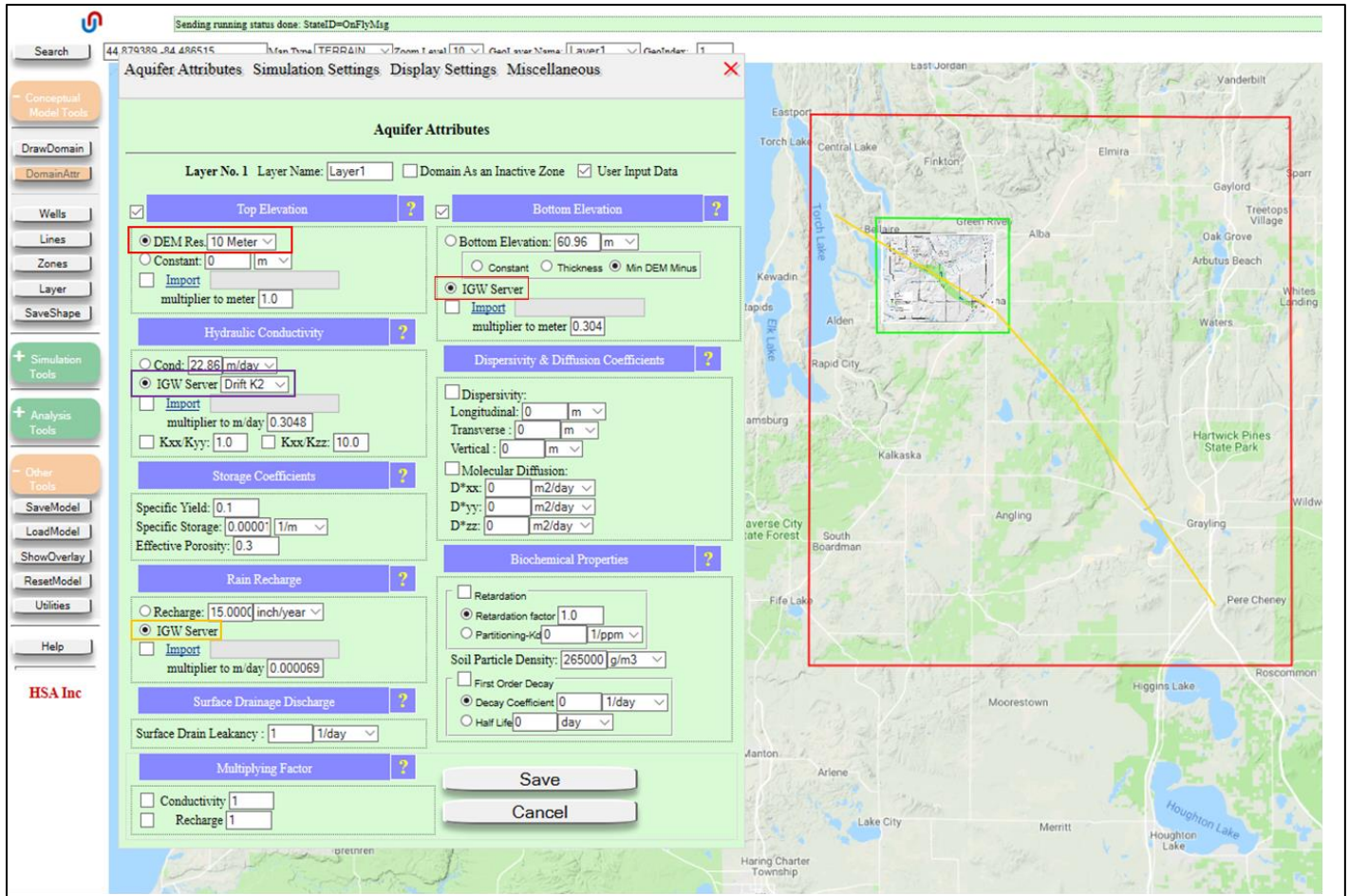


Figure 5. (Top) Assigning domain-wide attributes to the Regional Model: aquifer top boundary (Digital Elevation model); aquifer bottom boundary (bedrock surface raster); hydraulic conductivity (2D gridded raster file); and long-term mean recharge (2D gridded raster file). (Bottom) Conceptual cross-section of the model inputs/structure.

### 3: simulate the Regional model and examine a cross-section



- Run the model using the ‘Simulate’ tool (‘Simulation tools’ > ‘Simulate’) in steady-state mode (the default setting).
- After confirming the projection system, the flow field will be solved.
- Head contours and velocity vectors will be displayed in the regional model domain. Note that the number of vectors makes the display “crowded” in some areas (e.g., near the regional discharge areas).
- Reduce the vector size by navigating to the ‘Display Settings’ tab of the Domain Attributes menu and increasing the vector spacing from ‘1’ to ‘2’ pixels (see Figure 7)
- Use the ‘X-section’ Tool (‘Analysis Tools’ > ‘X-section’) to draw a cross-section from the regional recharge area to the regional discharge area. Click ‘ SaveShape’ (‘Conceptual Model tools’ > ‘SaveShape’) to finalize and display the cross-section in within the MAGNET modeling environment (see Figure 8).

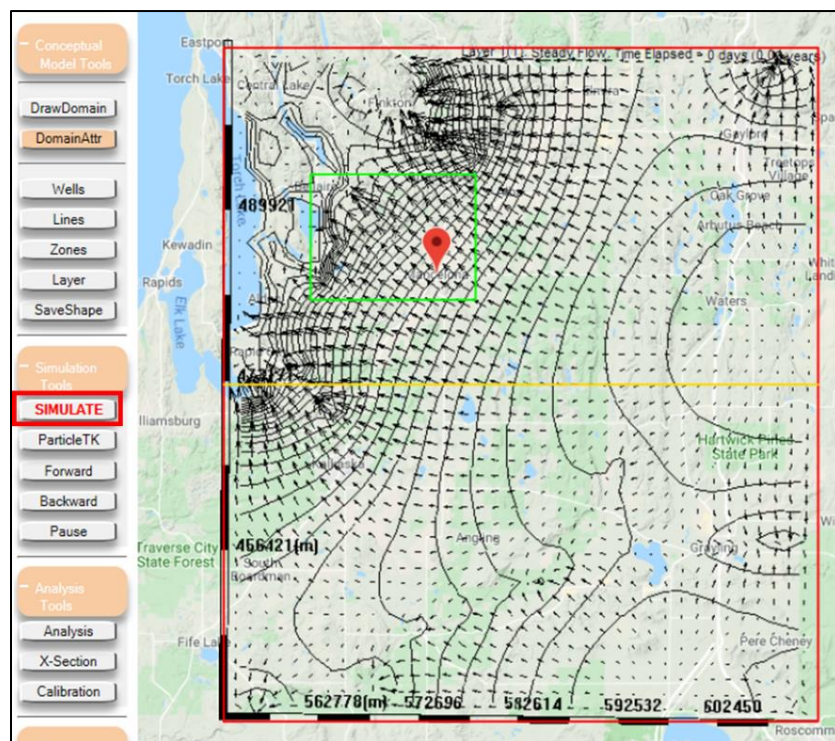


Figure 6. Model output after submitting the model for simulation. Contour lines represent head contours. Arrows represent groundwater velocity vectors, with the lengths indicative of the relative magnitudes of velocity.

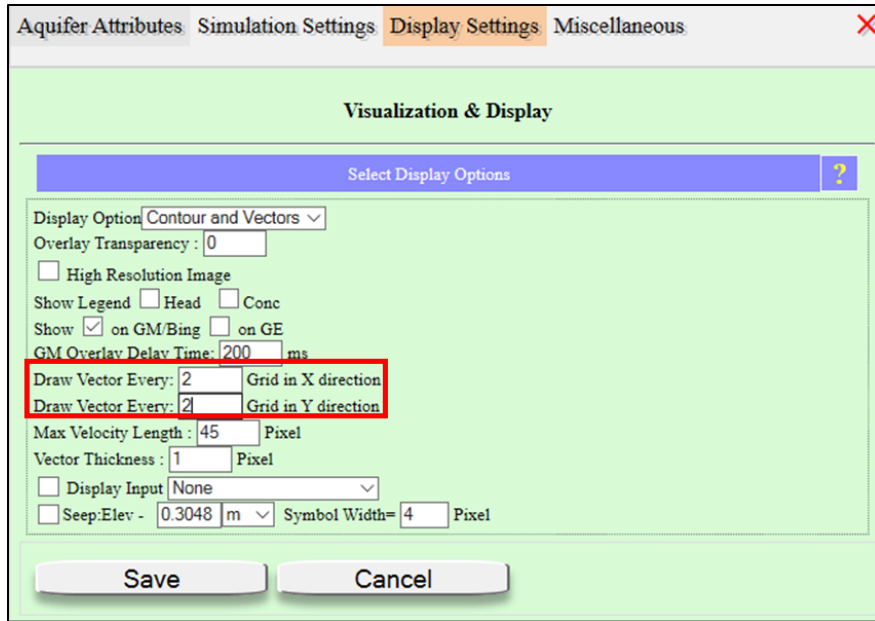


Figure 7. Adjusting the vector spacing in the Display Settings tab of the Domain Attributes menu.

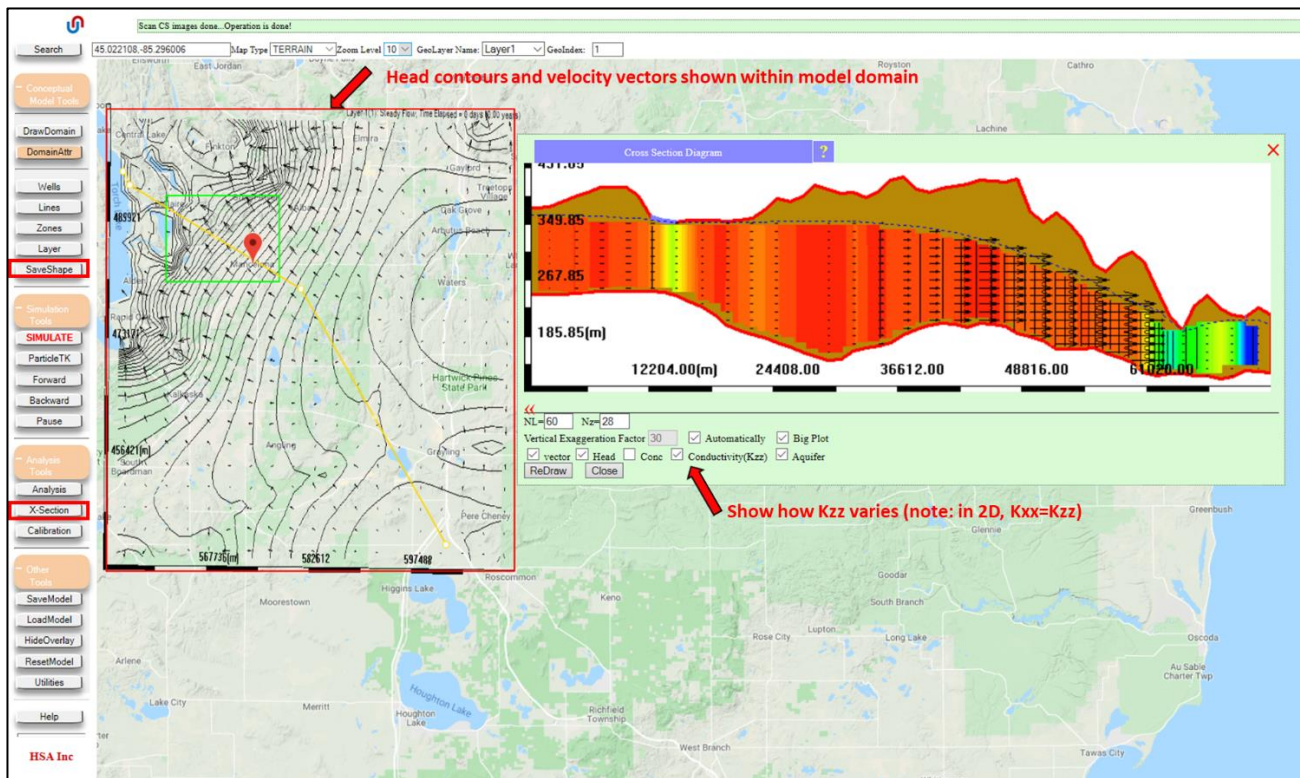


Figure 8. Updated plan view display and cross-section view. The variable colors in the cross-section represent variations in vertical hydraulic conductivity. (Note: in this 2D model,  $K_{xx} = K_{zz}$ ).

#### 4. Evaluate model performance (calibration)

- Compare Static Water Levels (SWLs) – measurements of water level in water wells at the time of installation (before pumping) – to the simulated model input by using the ‘Calibration’ tool (‘Analysis Tools’ > ‘Calibration’)
- Select ‘IGWServer’ as the data source in the Calibration Chart window.
  - Filter the data to only include wells confirmed to be screen in the surficial glacial aquifer and with installation dates after 2000.
- A comparison of simulated heads (y-axis) vs. observed heads (x-axis) will be shown in the chart display. Customize this chart:
  - Add confidence intervals of 1 standard deviation
  - Add a band-mean (moving window mean)
  - Reduce the data marker size
- Hit ‘ReDraw’ to update the chart display. Note that the model systematically underestimates head, especially in the range of head values of 250 to 375m.

Extract wells on the Server for calibration

Do not include rock wells

Temporal filter

Comparison, default settings

Comparison, customized settings

Show confidence intervals; reduce marker size; add moving-window average

Figure 9. The Calibration Chart and options/submenus. In this tutorial, Static Water Levels (SWLs) from a statewide water well database are used to evaluate the performance of the steady-state model. These data are noisy, which contributes to some of the spread seen in the data ‘cloud’. A lack of temporal or spatial resolution of the model also contributes to the data spread.

## 5. adjust K and recharge to improve model performance.



- To improve the match between model and data, adjust model inputs within a reasonable range, e.g., hydraulic conductivity and recharge.
- “Multipliers” can be applied to either the hydraulic conductivity data layer or the recharge layer. For example, a hydraulic conductivity multiplier of 1.1 results in the value of each cell in the raster to be multiplied by 1.1 (or an increase of 10%).
- Use the Domain Attributes menu to apply multipliers of 0.3 and 1.1 to the hydraulic conductivity layer and recharge layer, respectively (see Figure 10).

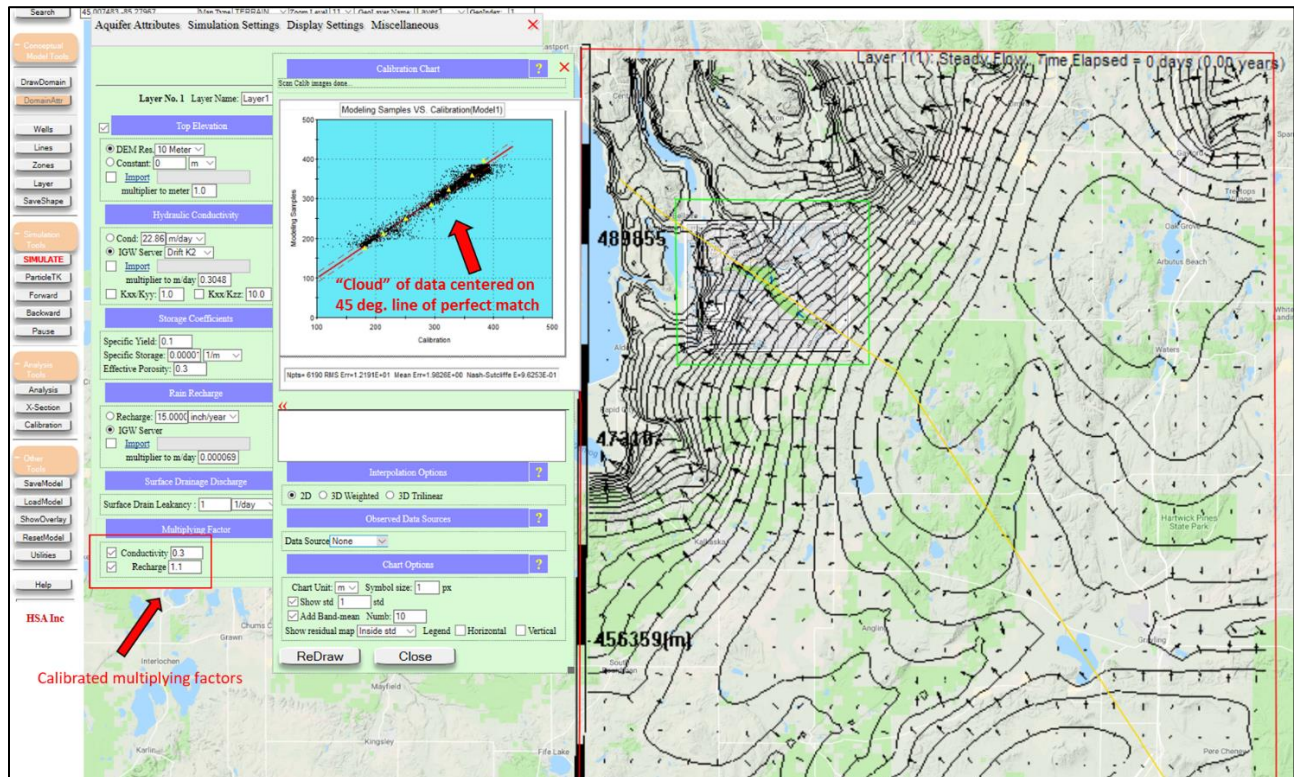


Figure 10. Calibrated regional model results. Note that both the cloud of data and the band-mean is centered on the 45 deg. line of perfect match.

## 6. make the local model active and submit the submodel for simulation.

- Use the results of the regional model to provide boundary conditions for the local model. Specifically, the regional model provides a spatially-variable head values along the perimeter of the local model domain.
- First, edit the local model zone.
  - Use the ‘Geometry unlocked’ tool (‘Utilities’ > ‘Geometry unlocked’) to visualize the nodes of the local model zone.
  - Click any of the nodes to launch the Zone Attributes menu.
  - Check the box next to ‘Submodel domain’ and make sure the ‘Active Option’ is selected (see Figure 11).

- Then, navigate to the Simulation Settings tab of the Domain Attributes menu and check the box next to Boundary Conditions from Parent Model (see Figure 12). This utilized the last simulation result – the regional model flow field – to derive boundary conditions for the local model.
  - Note: the user can save the latest simulation results to be used as boundary conditions by selecting ‘Latest Model Zipped File’ from the ‘SaveModel’ menu. (‘Other Tools’ > ‘SaveModel’ > ‘Latest Model Zipped File’).
  - This file can then be uploaded and selected as the boundary condition for any subsequent simulation.
- Submit the job for simulation and view the results. Draw a new cross-section to visualize the flow patterns along the direction of the TCE plume (see Figure 13).

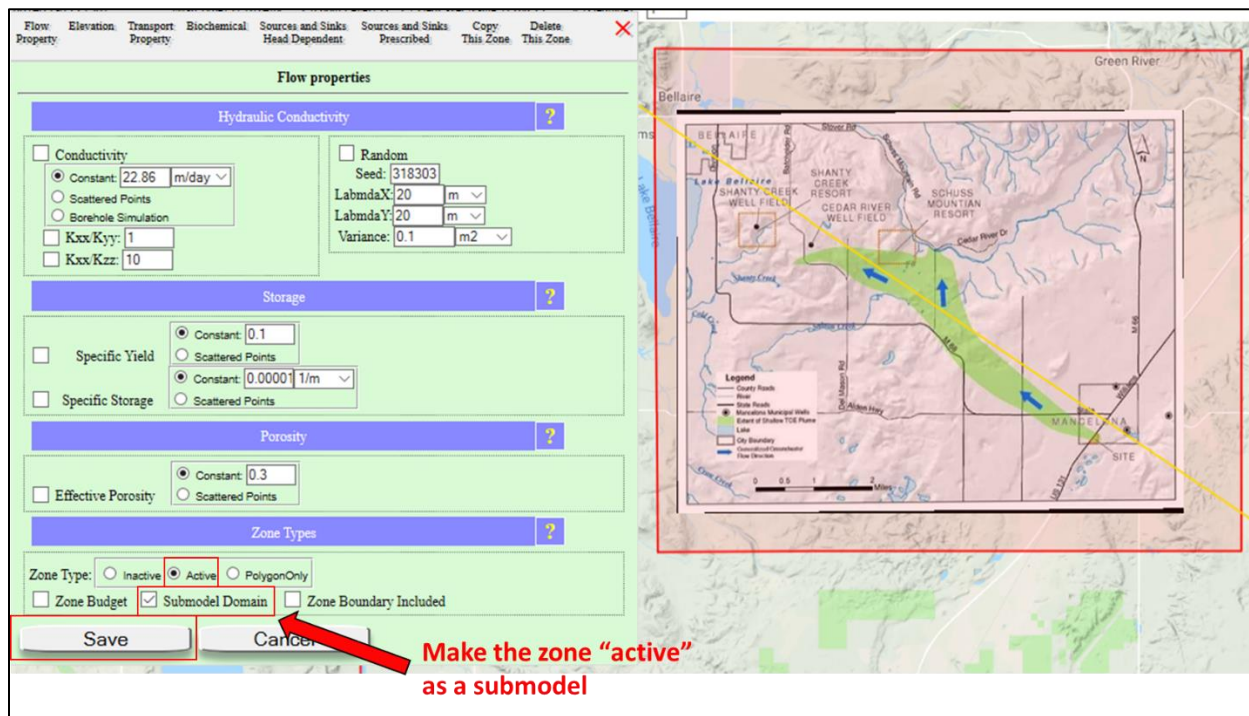


Figure 11. Making the submodel, or Local Model, active in the Zone Attributes menu. This menu is accessed by using the ‘ZoneAttr’ tool (‘Conceptual Model Tools’ > ‘Zones’ > ‘ZoneAttr’), or by unlocking the geometry (‘Utilities’ > ‘Geometry unlocked’) and clicking one of the nodes of the Local Model zone.

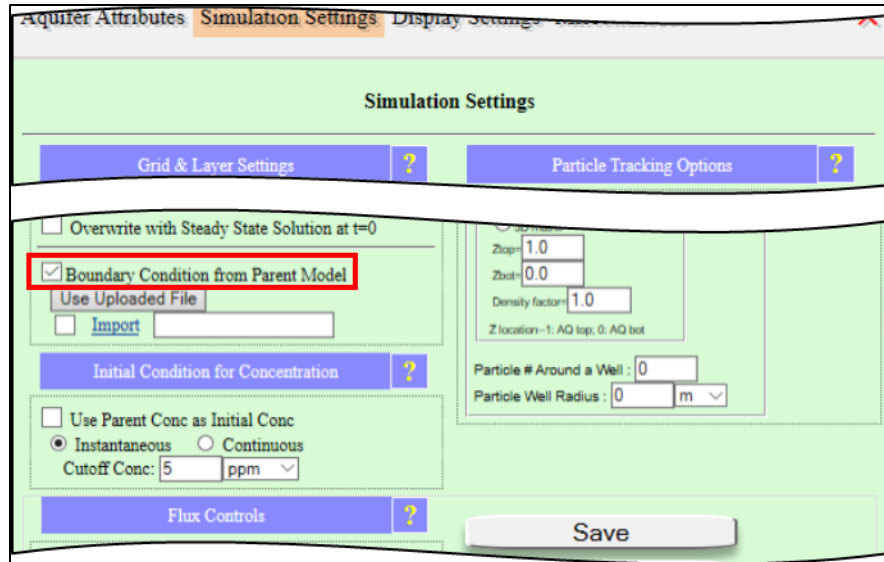


Figure 12. Assigning the boundary condition (head/flux values) from the Parent Model (Regional Model) in the Simulation Settings tab of the Domain Attributes menu.

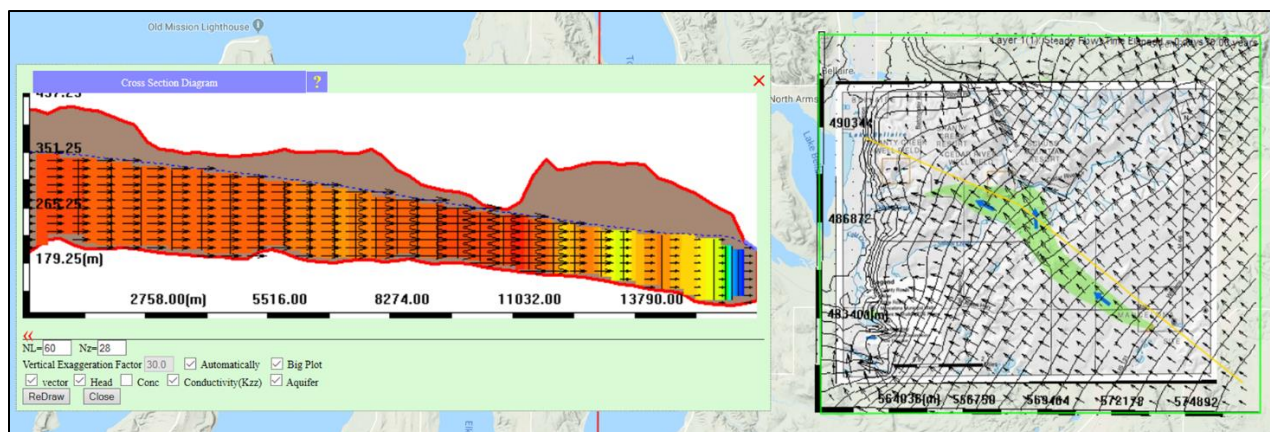


Figure 13. Local model simulated results.

## 7. Adding a stream feature

- A salient feature of the site is the Cedar River, situated north-northeast of the TCE plume and flowing roughly east to west-northwest, eventually draining into Lake Bellaire.
- Add the Cedar River as a conceptual feature using the 'DrawLine' tool. ('Conceptual Model Tools' > 'Lines' > 'DrawLine' tool).
  - Point-and-click to add stream vertices, using the map display or site map as a guide.
  - Use 'SaveShape' to finalize the stream feature.
- The Line Attributes menu will launch. Assign the Cedar River line feature as a two-way, head dependent boundary condition, allowing water to be exchanged between the stream and aquifer based on the relative difference hydraulic head (Figure 14).
  - Assign the stream stage to follow the aquifer top (DEM).
  - Assign the stream bottom elevation to be 1m below the stream stage.



- Assign a stream leakance (hydraulic conductivity per unit thickness of the streambed) of 5 m/day.

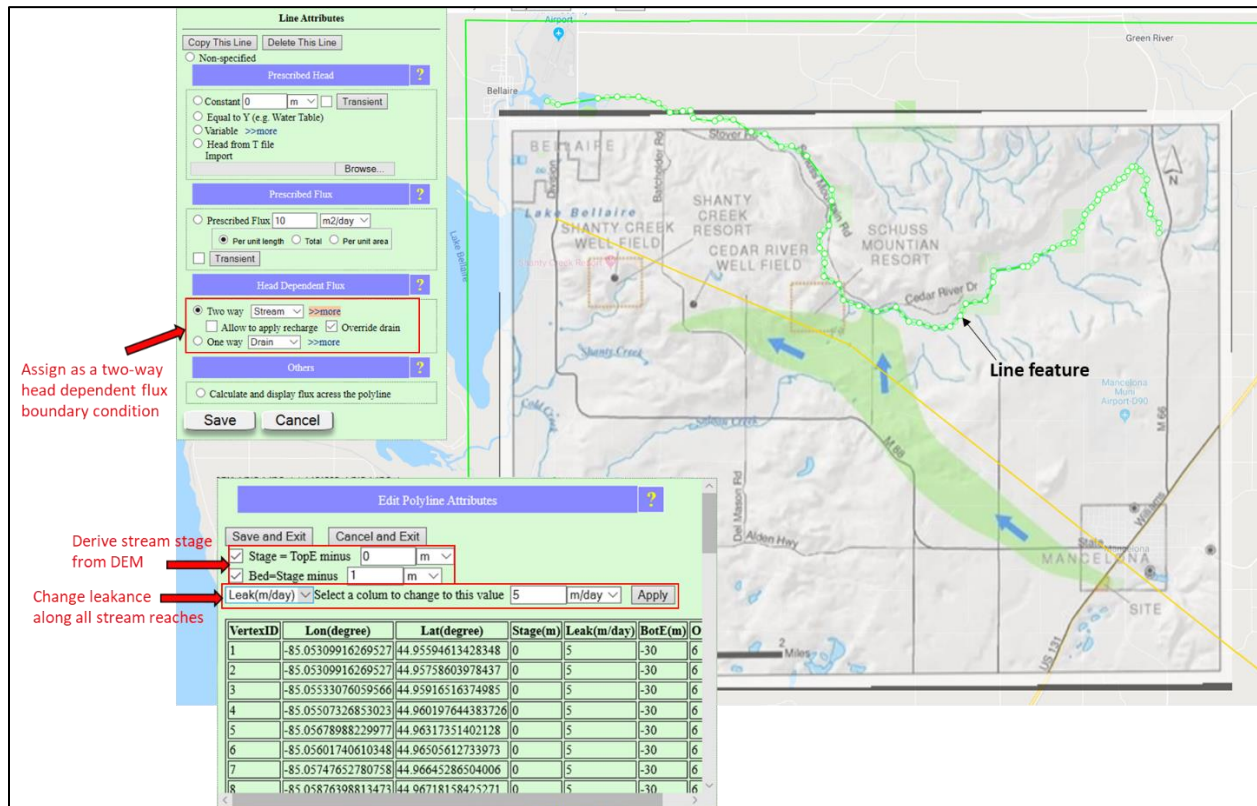


Figure 14. Adding a stream (line) feature to the Local Model. When using the 'DrawLine' tool, cursor will become a cross-hair, allowing for placement of polyline vertices with single-clicks of the LM button. Once the last vertex has been placed, LM click the 'SaveShape' button to finalize the polyline.

## 8. Add source of contamination, add a monitoring well downstream

- Add the source of contamination at the property formerly used by Wickes Manufacturing.
- Zoom into the site (see Figure 15), then use the 'ZonePoly' tool to draw a zone at the site. ('Conceptual Model Tools' > 'Zones' > 'ZonePoly')
- Click 'SaveShape' to finalize; the Zone Attributes menu will open. Navigate to the 'Source and Sinks Prescribed' tab, then assign the feature as a continuous source with a concentration of 1000 ppm (parts per million).
- Use the 'Well' tool ('Conceptual Model Tools' > 'Well') to add a monitor well downstream from the contamination site.
- Check the box next to 'Monitoring Well' (see Figure 16).

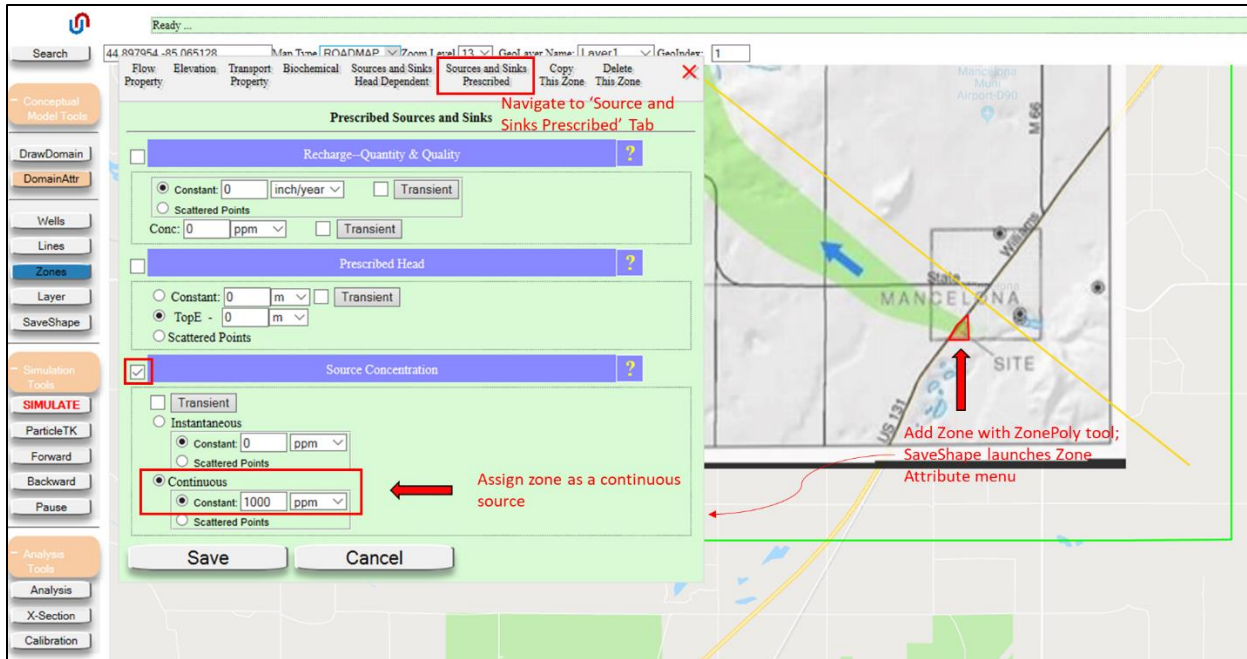


Figure 15. Adding a source of contamination at the spill site using the 'ZonePoly' tool and the Source and Sinks Prescribed tab of the Zone Attributes menu.

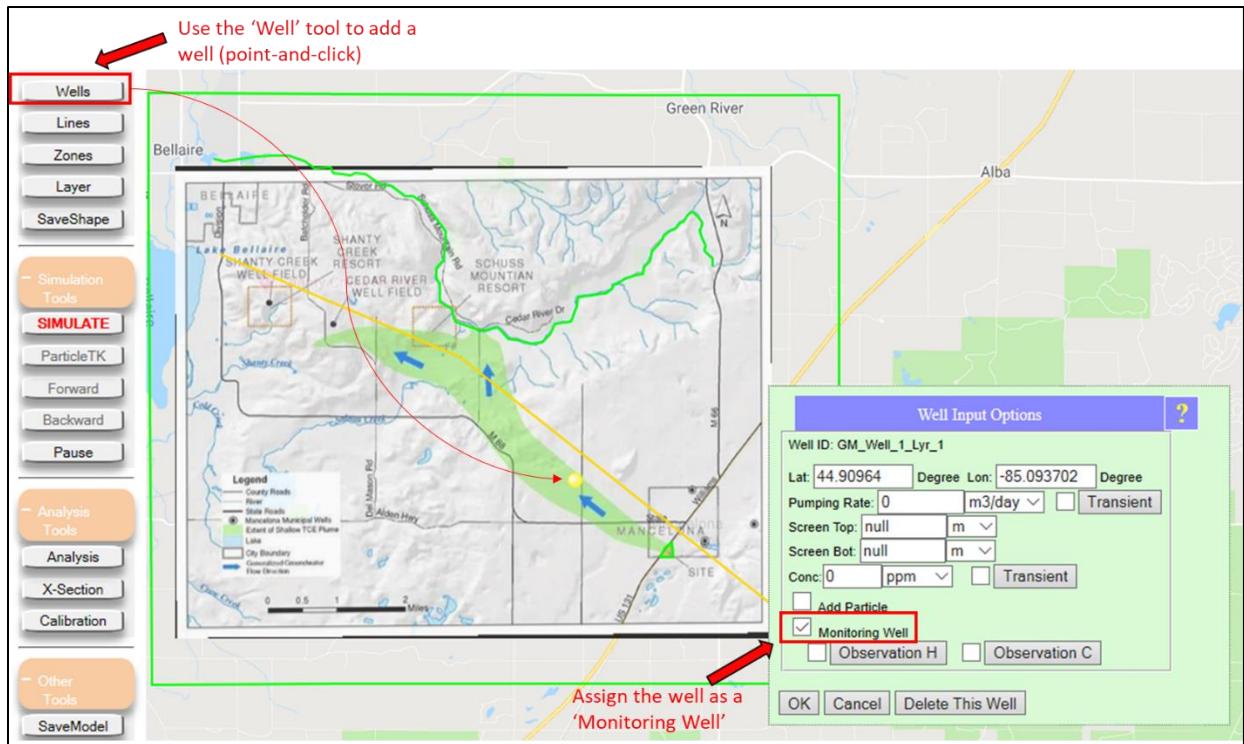


Figure 16. Adding a monitoring well using the 'DrawWells' tool ('Conceptual Model Tools' > 'Wells' > 'DrawWell') and the Well Input Options menu.

## 9. simulate and compare simulated transport to traditional delineation

- Before simulating the flow and contaminant transport, assign boundary conditions from the parent model in the Domain Attributes menu (by default, 'Boundary Conditions from Parent Model' is unchecked after each simulation).
- Submit local model for simulation.
- The areal extent of the contamination and the head contours and velocity vectors will be shown in map display. The simulation is steady in the sense that the flow field does not change with time, but the plume is allowed to spread with time based on the simulated steady flow field.
- Use the 'DisplayCharts' tool to show the monitoring well, cross-section view and mass balance chart in the MAGNET modeling environment ( 'Analysis Tools' > 'Analysis' > 'Display Charts'). See Figure 17.
- Check the model performance with the 'Calibration' tool.

Note that the local model performs reasonably well: there is a good match in the calibration chart; and the plume direction and extent is generally consistent with the plume delineated with traditional hydrogeological field methods.

You may also notice that the simulated plume is not "pulled" to the west as field data/site map suggests in does in reality. This is because the model does not simulate the Cedar River well field near Shanty Creek Resort. Wells can be added to the model using the 'DrawWell' tool ('Conceptual Model Tools' > 'Wells' > 'DrawWell') – see Figure 16.



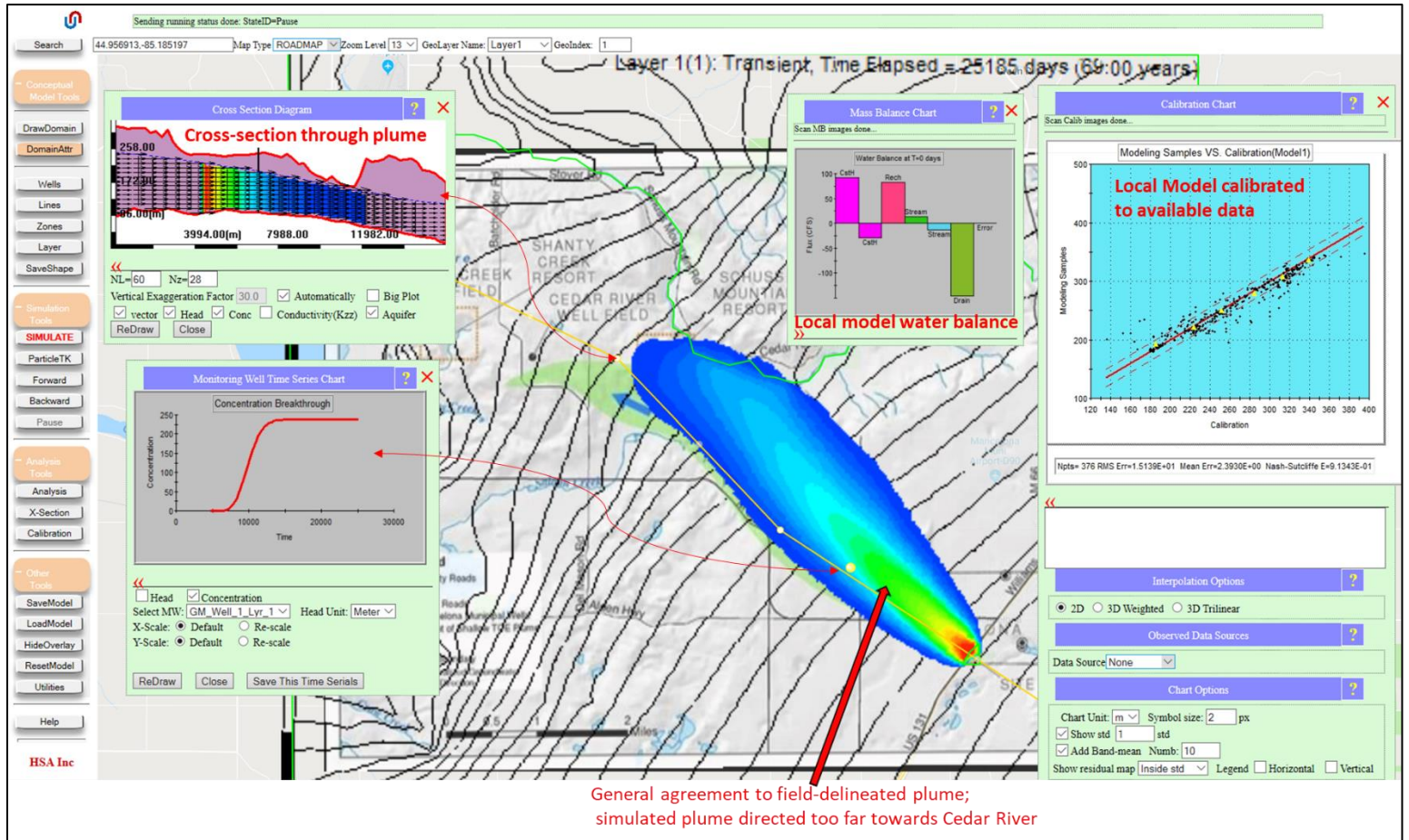


Figure 17. Simulated model results, including the plume extent (plan and cross-section views), the concentration breakthrough curve at the monitoring well, the aquifer water balance, and the local model calibration chart. The charts/plots are displayed using the 'DisplayCharts' tool ('Analysis Tools' > 'Analysis' > 'DisplayCharts').

## 10. Add purge wells to remove contamination

- Add a purge well to the model to remove the contaminated groundwater from the surficial aquifer.
- Use the 'DrawWell' tool to add an extraction well downstream from the source of contamination.
  - Assign an aggressive pumping rate: 2000 GPM. (Note that it should be a negative value for pumping/extraction.)
- Submit the model for simulation (after ensuring the boundary conditions are coming from the parent model). Visualize the plume migration now that the purge well has been added (see Figure 18).

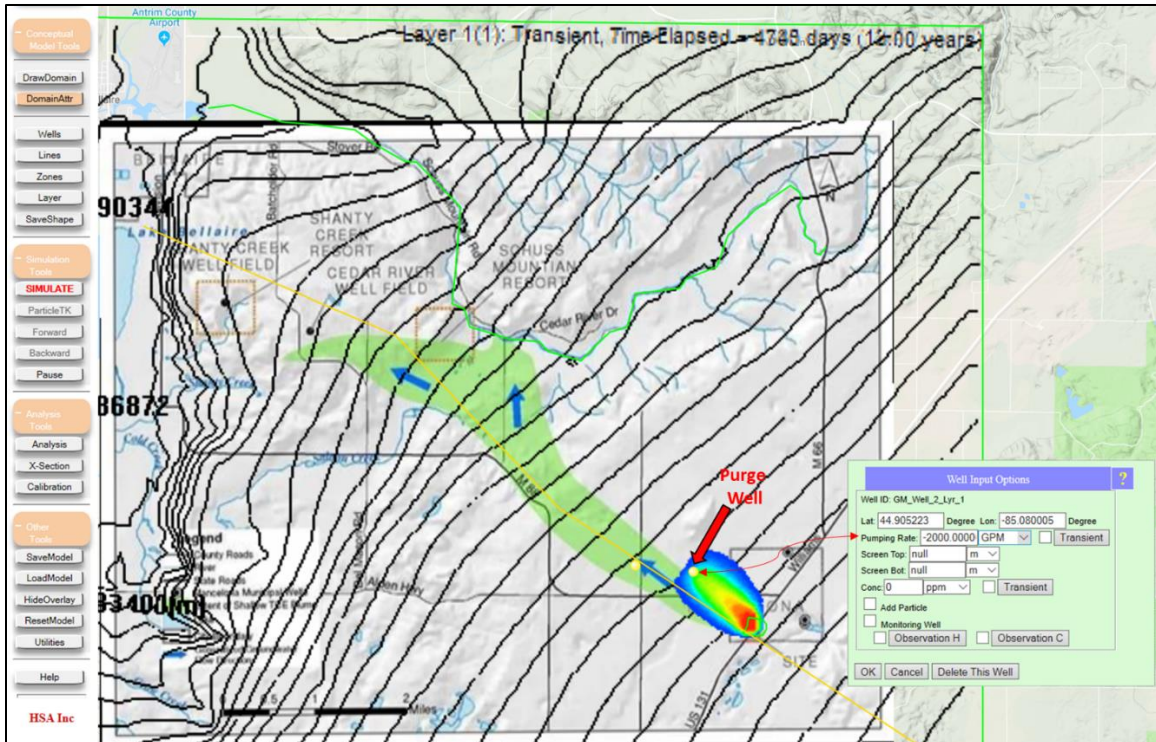


Figure 18. Simulated plume transport with a purge well added downstream of the spill site. Note the much smaller extent of the plume and the bended contour lines near the purge well.

## 11. add vertical details – multiple computational layers and the source at the surface

The flow and plume transport has been simulated in two dimensions (XY) up to this point, with the assumption that the plume is perfectly mixed in the vertical direction. In reality, the contamination enters the groundwater system from above and is not evenly mixed in the vertical direction. ‘Computational layers’ will be added to resolve the head and concentration variability in the vertical directions.

- First convert the ‘source’ zone from a continuous source to a surface source by opening its Zone Attributes window and navigating to the ‘Source and Sink Prescribed’ tab (Figure 19).
  - Uncheck the box next to ‘Source Concentrations’.
  - Check the box next to ‘Recharge – Quantity & Quality’.
  - Enter a constant rate of infiltration (recharge) of ‘10’ inch/year.
  - Enter a source concentration (‘Conc’) of 1000 ppm.
- Sub-divide the saturated aquifer thickness into multiple (eight) vertical computational layers (Figure 20).
  - Navigate to the ‘Simulation Settings’ tab of the Domain Attributes menu.
  - Check the boxes next to ‘Water Table as Top’ and ‘Number of SubLayers’ and enter ‘8’ for the number of sublayers. This tell MAGNET to subdivide the saturated thickness into 8 layers of equal thickness, using the water table from the 2D simulation as a guide.

- Double-check that the boundary conditions are derived from the Parent (Regional) model, then re-submit the model for simulation using the 'SIMULATE' button. Observe the plume using a cross-section (Figure 21).

Note that the simulated plume now occupies the upper ~1/3 of the aquifer rather than being vertically-mixed as was seen with the 2D model.

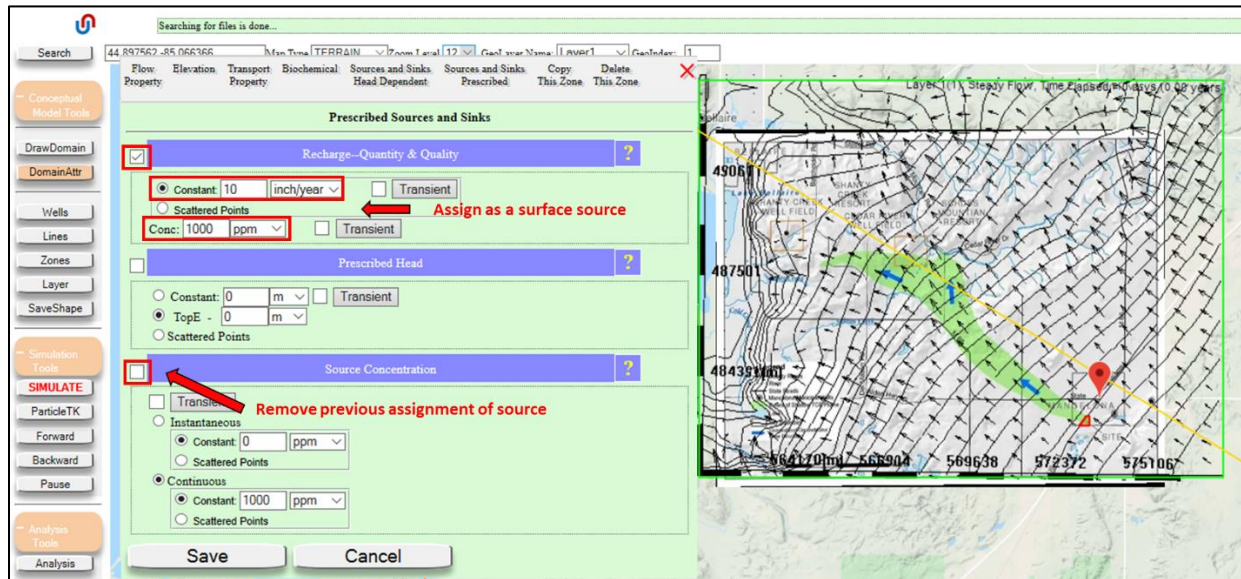


Figure 19. Changing the type of contamination site represented in the model – from a perfectly (vertically) mixed source, to a surface source entering the groundwater system from above.



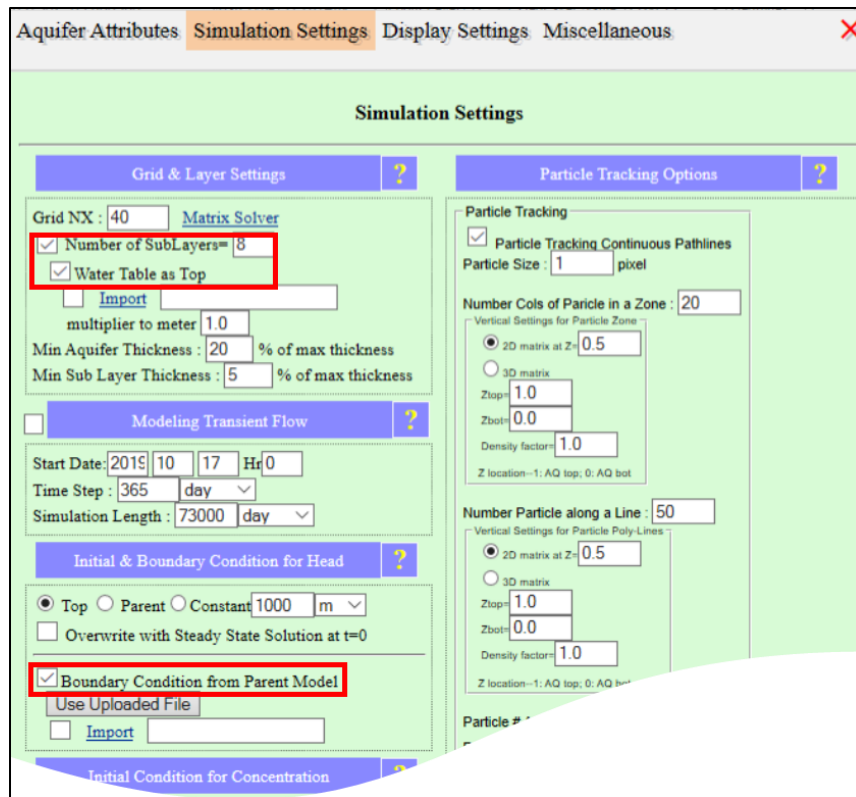


Figure 20. Adding vertical computational layers to the Local model. MAGNET will subdivide the saturated thickness into 8 layers of equal thickness, using the water table from the 2D simulation as a guide. Make sure the box next to 'Boundary Condition from Parent Model' is checked each time you want to run the submodel (Local Model).

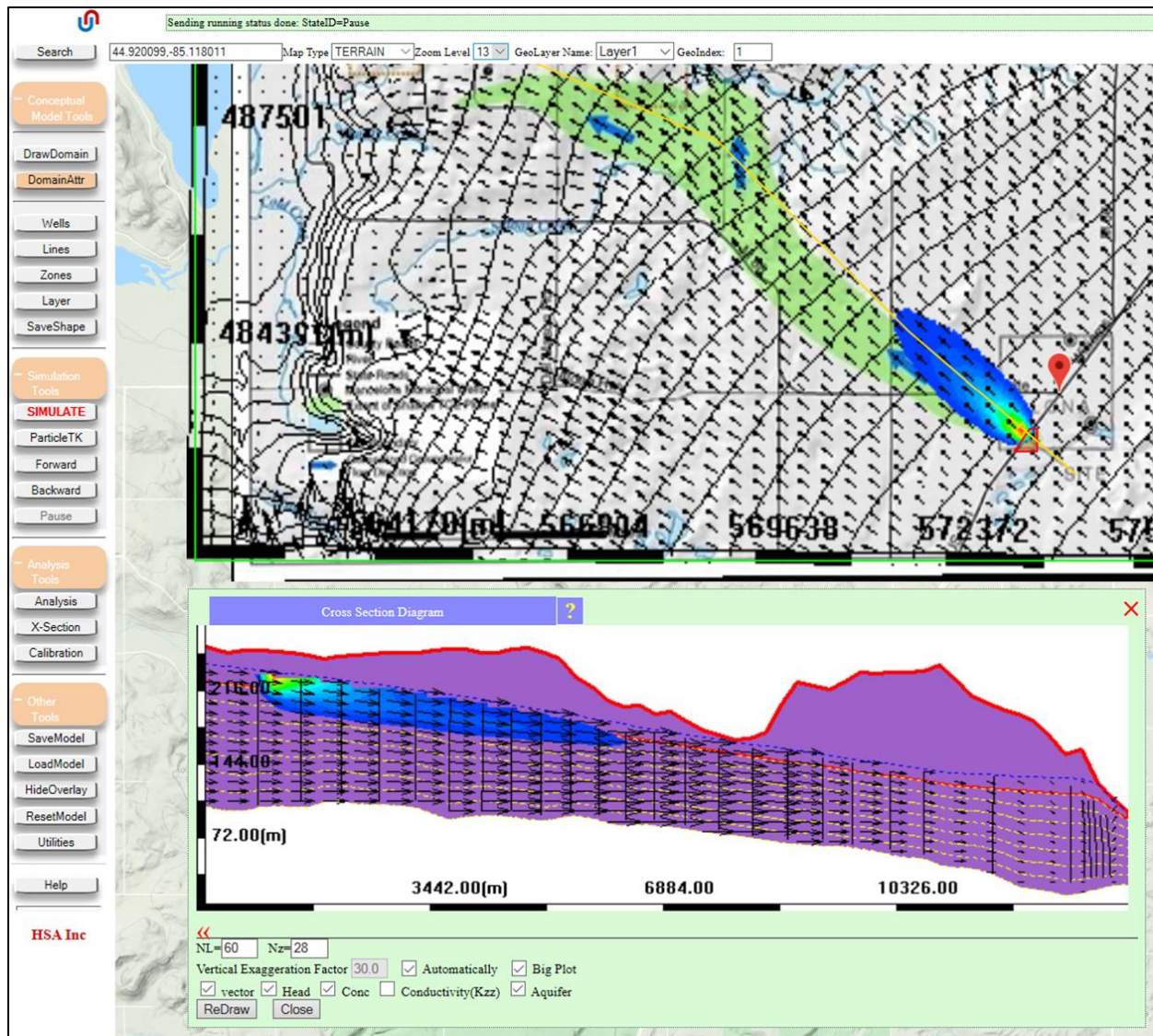


Figure 21. Simulated model results, with vertical details resolved by the new submodel. Note that that the simulated plume now occupies the upper ~1/3 of the aquifer rather than being vertically-mixed as was seen with the 2D model.