TUTORIAL 6 – Capturing Vertical Dynamics

==========

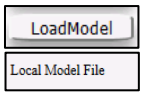
This tutorial demonstrates the use of MAGNET to create a groundwater model with multiple computational layers within an aquifer unit *to provide greater vertical resolution of processes. First, a 2-dimensional model is created, which is then converted to a pseudo-3-dimensional model by "following the water table" to vertically sub-divide the aquifer into multiple computational layers.*

The flow model created in a previous tutorial (Tutorial 2 – Nested Models) is used to illustrate this example.

Tutorial 10 explains how to add an entirely new aquifer layer (aka, a hydrostratigraphic unit, a geological layer, a conceptual layer, etc.), which has its own set of zones, lines, attributes, computational layers, etc.

"pseudo-3D" Model – Fort Custer, Michigan

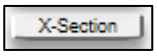
- *Step 1* – load and simulate the regional model from a previous tutorial (see Tutorial 1), then load the nested submodel (Figure 1) and apply 'Boundary Conditions from Parent Model' in the Default Input Model Input Parameters and Display Options menu (see Tutorial 2).



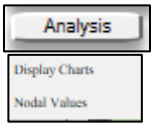
- *Step 2* – Submit the model for simulation*.

- *Step 3* – Open Simulation Settings tab in the Default Input Model Input Parameters and Display Options menu and check the box next to 'SubLayers=' and specify the number of vertical computational layers to use (this example uses 3 sublayers).

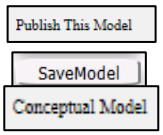
- *Step 4* – Still within the Simulation Settings tab, check the box next to 'Water Table as Top'*, then save the changes by clicking 'OK'.
- *Step 5* – Draw a cross-section in an area of interest (see Figure 2).



- *Step 6* – re-submit the model for simulation.
- *Step 7* – Click the 'Analysis' button, then select 'Display Charts' from the submenu. This launches a number of windows for analyzing modeling results, including the Full Cross-Section chart and the 3D forms chart ** (see Figure 3).



- *Step 8* – Publish/save the final submodel model for future use



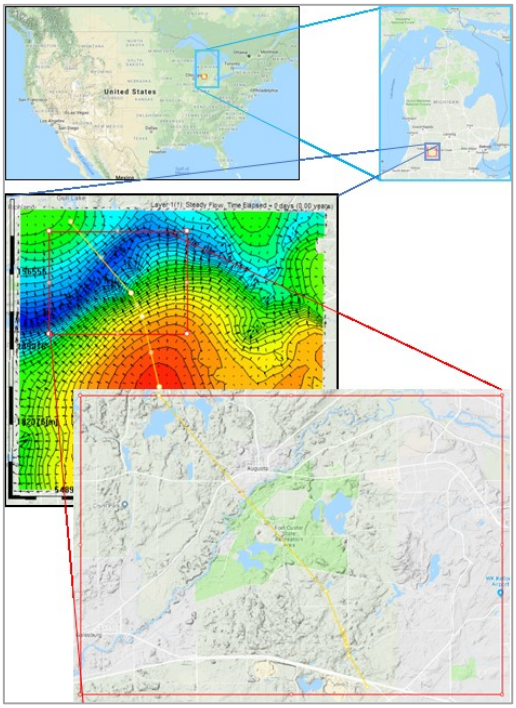


Figure 1 (right): Flow model, loaded into the MAGNET modeling environment and simulated with results displayed.

Additional Details:

- *The 2D model result provides an initial water table for further vertical discretization. The default setting is to use the land surface as the top boundary for sub-dividing. It is also possible to upload a saved water table. For more details, see the Real-time help buttons for Save Model and Grid and Layer Settings (under Simulation Settings in the Default Model Input Parameters and Display Options menu).
- It is best to incrementally increase the number of layers, e.g., first simulate 2D flow patterns, then sub-divide into 2 or 3 layers, then further sub-divide into 4 or 5 layers, and so on.
    - 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.
- Relevant sections in MAGNET manual:
    - Step 1: Section 7.1
    - Step 2: Section 5.1
    - Steps 3-4: Section 3.2
    - Step 5: Section 5.4

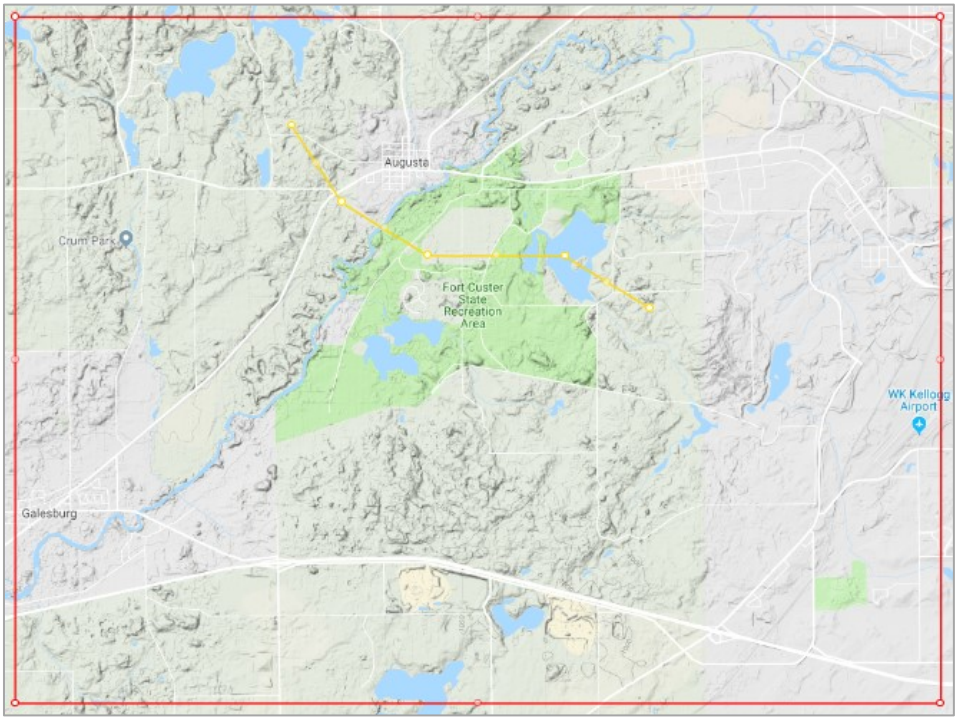


Figure 2: Drawing a cross-section in an area of interest to visual vertical flow dynamics.

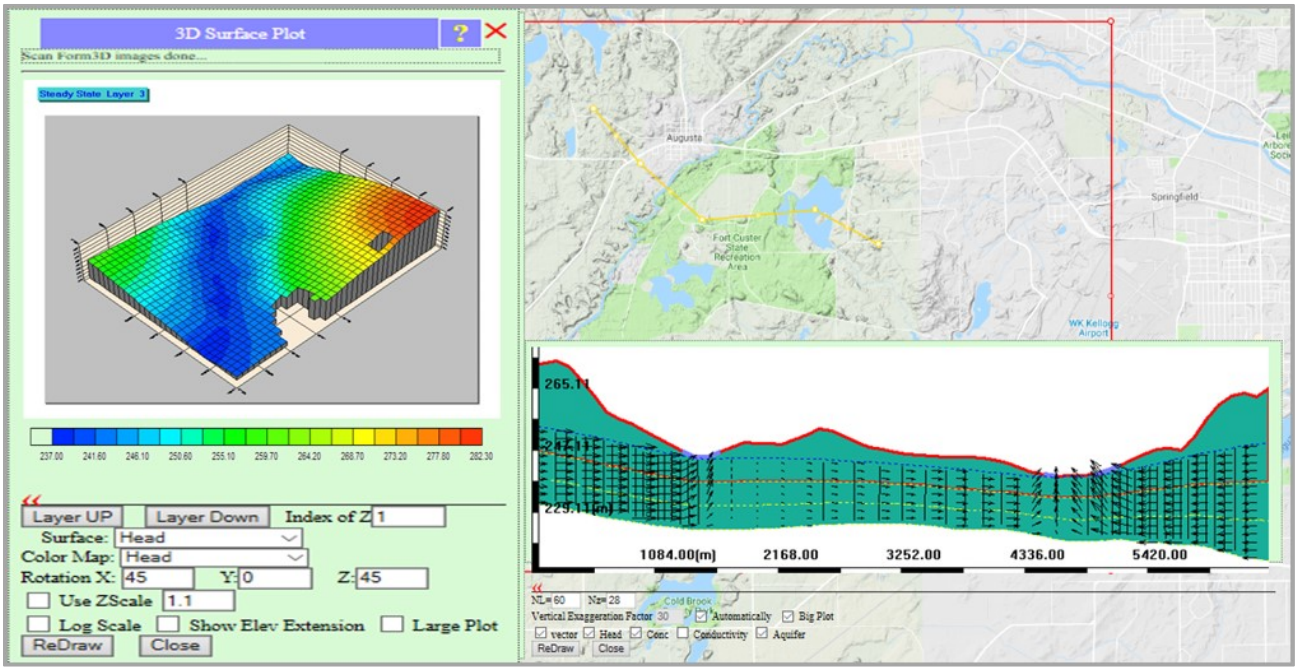


Figure 3: Results of the 3-layer model. The 3D form chart is shown on the left, with the head surface and color map of the bottom-most layer shown. Note that a few dry cells may occur if the aquifer thickness and other aquifer parameters, e.g., recharge, conductivity, are not consistent. The cross-section in the bottom-right shows the water table (blue dotted line), seepage areas (blue thick lines along the land surface, and velocity vectors for three different computational layers (differentiated with dotted yellow and red lines). The 3D model clearly shows groundwater upwelling to the major surface water bodies through which the cross-section intersects, i.e., the large lake at the eastern end of the cross-section and the Kalamazoo river near the western end.