TUTORIAL 1 – Steady 2D Flow Modeling

==

This tutorial explains how to quickly create and visualize a 2D, steady-state flow simulation using A) spatially-explicit inputs (e.g., recharge, conductivity, aquifer bottom) from the MAGNET Data Center; and B) single, effective values as inputs (pragmatic for sites currently lacking detailed inputs on the Data Center). This Tutorial also shows C) how to utilize the MODFLOW solver engine rather than the default MAGNET engine (from Interactive GroundWater, or IGW, source code).

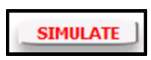
This tutorial assumes the user has created a MAGNET account, which is required to submit jobs for simulation, save models, etc.

A – Model with Data Center Inputs

- *Step 1* – Zoom to the area of interest and delineate regional model domain (Figure A1).

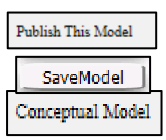
- *Step 2* – assign default parameters from the Server for the model domain (Figure A2).

- *Step 3* – Submit job for simulation.



- *Step 4* – Visualize flow patterns in plan and cross-section view (Figure A3). Use the 'X-section' tool to draw a new cross-section.

- *Step 5* – Publish/save the model for future use.



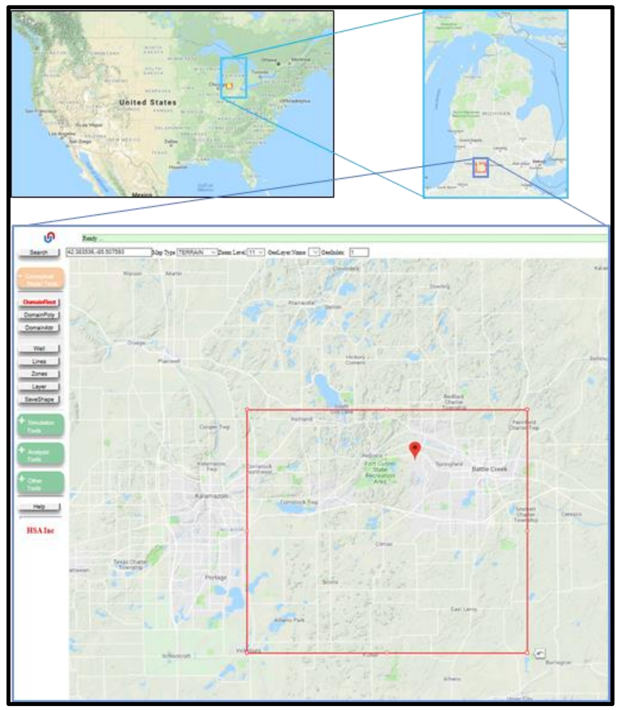


Figure A1: Regional model domain (red rectangle) for the area of interest: Fort Custer State Recreation Area near Kalamazoo, Michigan, U.S.A

<u>Additional Details</u>:

- At the current time, spatially-explicit representations of recharge, conductivity, and bottom aquifer elevation are only available for the State of Michigan, U.S.A. (Expanded coverage coming soon!)
- 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. The general idea is to: first create and simulate a steady-state model, then create a smaller, nested model (or series of models) that derives its/their boundary conditions from the larger model.
    - See Tutorial 2 – Nested Modeling; and Tutorial 11- Zone-based Hierarchical Modeling

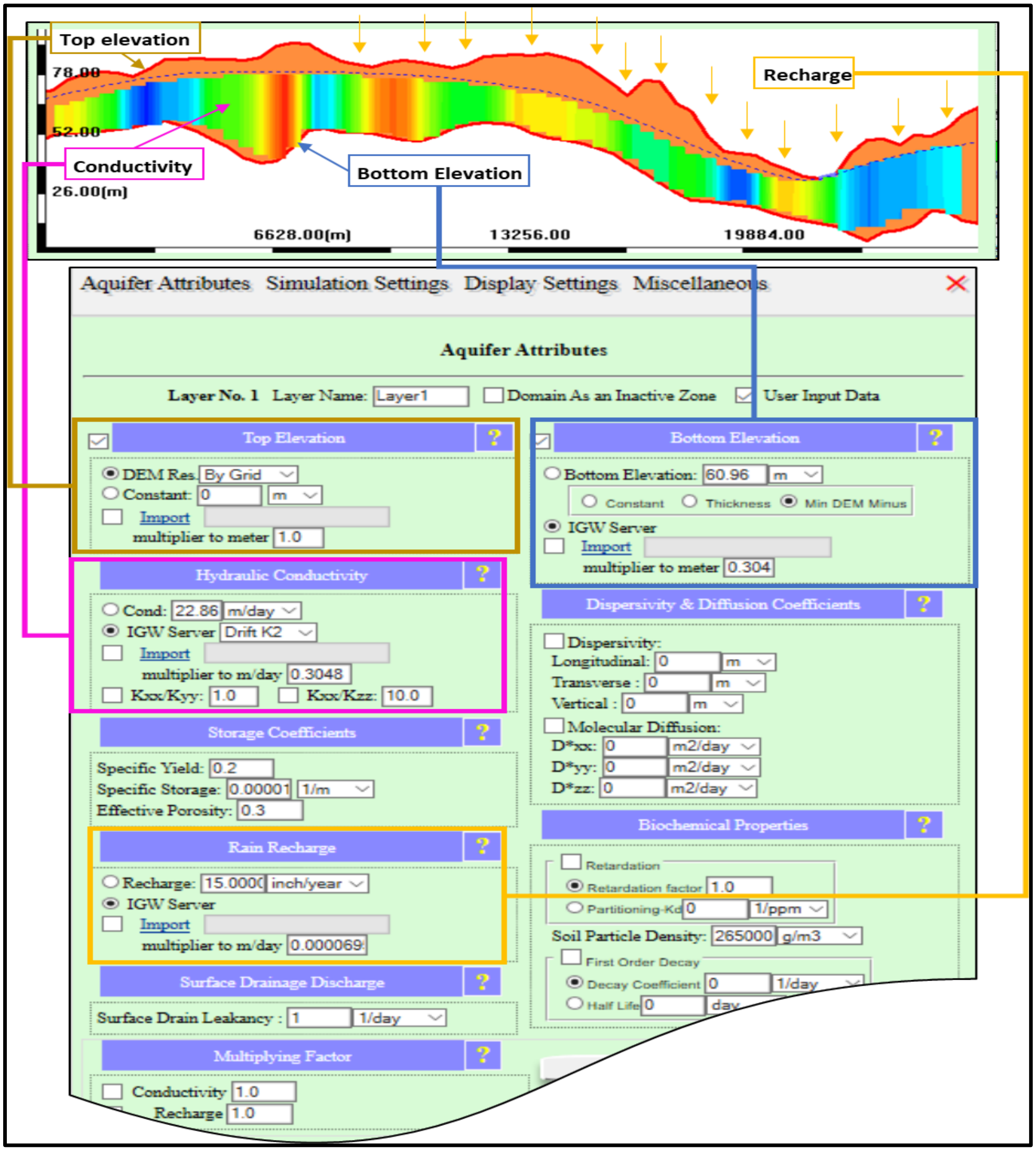


Figure A2: Assigning input parameters from the Server.

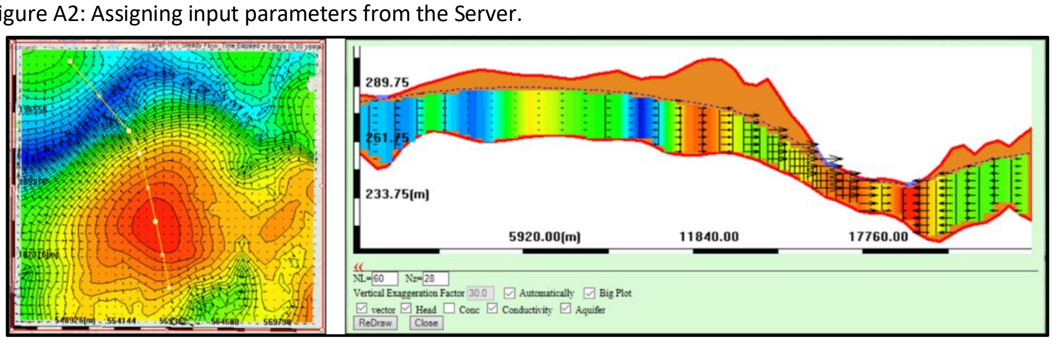
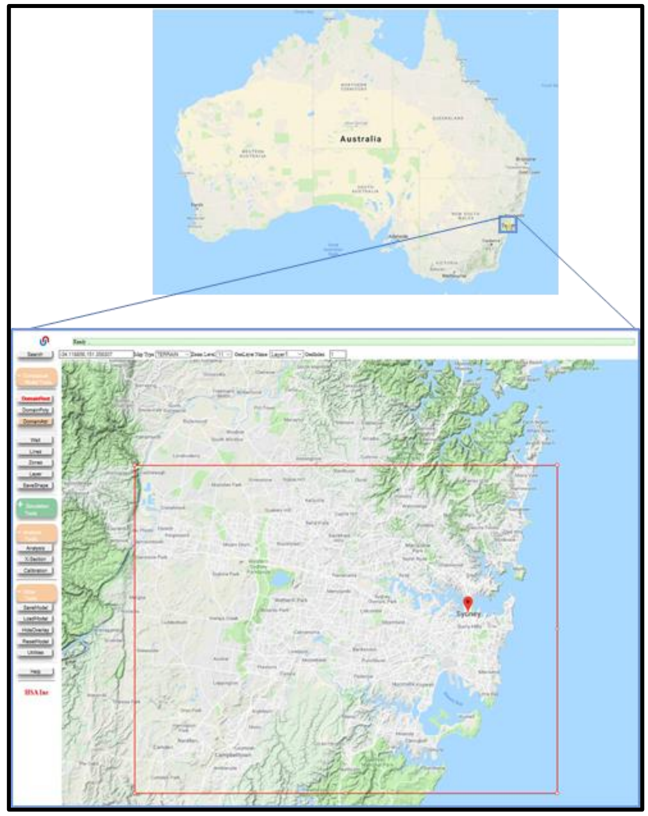


Figure A3: Flow patterns for the Michigan-based model: (left) plan-view (with the color map indicating head) and (right) cross-section view (with the color map indicating conductivity and the arrows representing velocity vectors).

B – Model with Effective Value Inputs



- Step 1 – delineate the regional model domain (Figure B1).

- Step 2 – assign input parameters as single, effective values* (Figure B2).

- Step 3 – Submit job for simulation.

- Step 4 – Visualize flow patterns in plan and cross-section view; examine model inputs/results at a node location (Figure B3).

- Step 5 – Publish/save the model for future use.

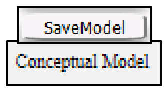


Figure B1: Regional model domain for location of interest: greater area of Sydney, Australia.

Additional Details:

- *Spatially-explicit values of recharge, conductivity, and bottom aquifer elevation are not currently available outside of Michigan. However, datasets will be integrated as they become available and of acceptable quality. Hence the use of single, effective parameters for models around the globe as a “starting point” for further customization.
- Users can also load spatially-explicit raster files for K, recharge, bottom elevation (even DEM too)

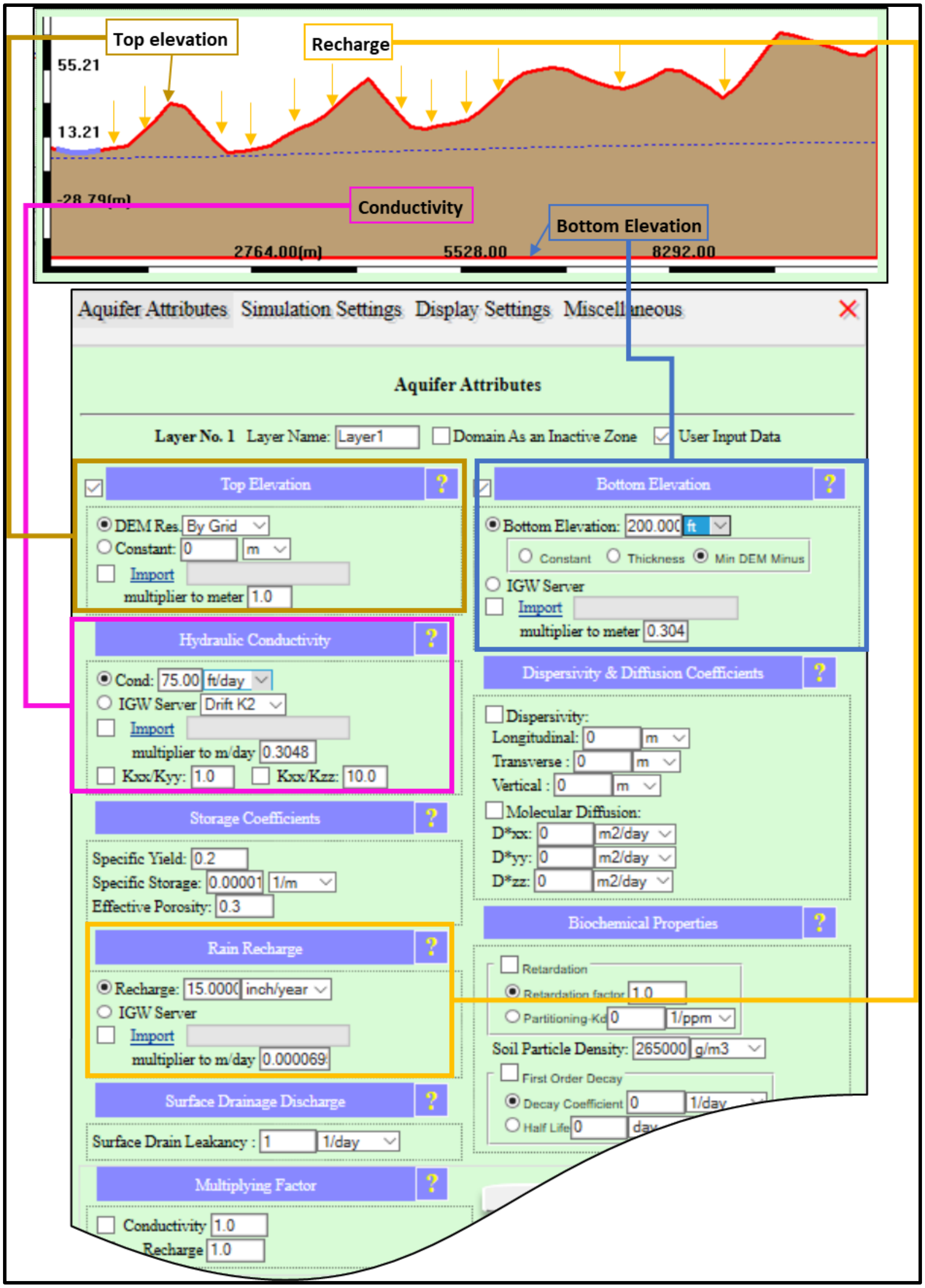


Figure B2: Assigning input as single, effective values using the Default Model Input Parameters and Display Options menu (with the exception of DEM).

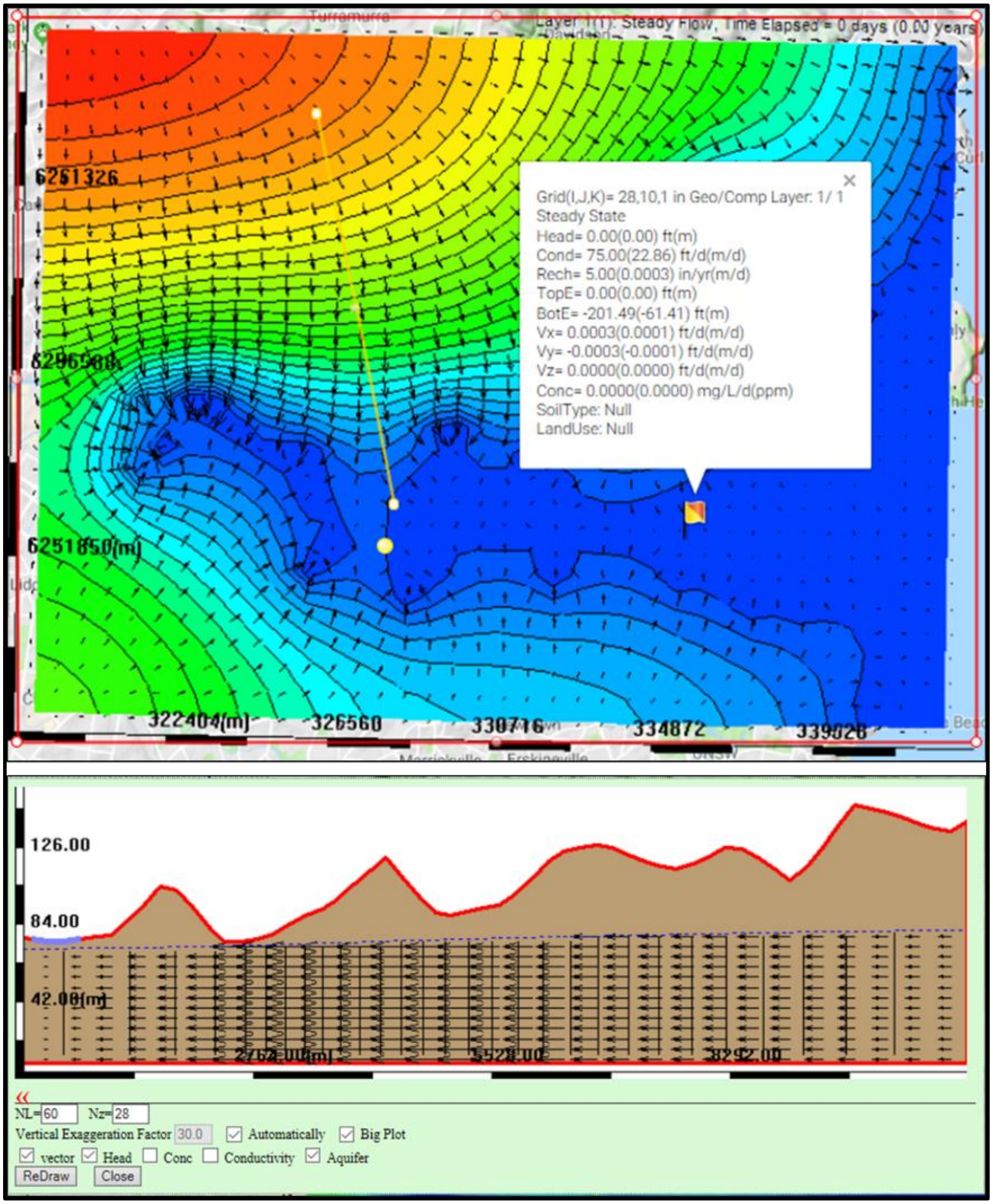
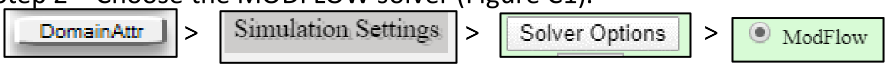


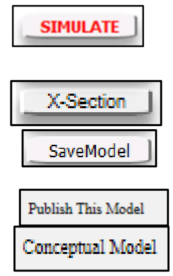
Figure B3: Flow patterns for global network model: (top) plan-view (with the color map indicating head) and model inputs/results at a node; and (right) cross-section view (with the color map indicated conductivity and the arrows representing velocity vectors).

C – Using the MODFLOW Solver

- Step 1 – follow steps 1-2 above to select an area and input parameters for either type of 2-D flow model.
- Step 2 – Choose the MODFLOW solver (Figure C1).



- Step 3 – Submit job for simulation.
- Step 4 – Visualize flow patterns in plan or cross-section view
- Step 5 – Publish/save the model for future use.



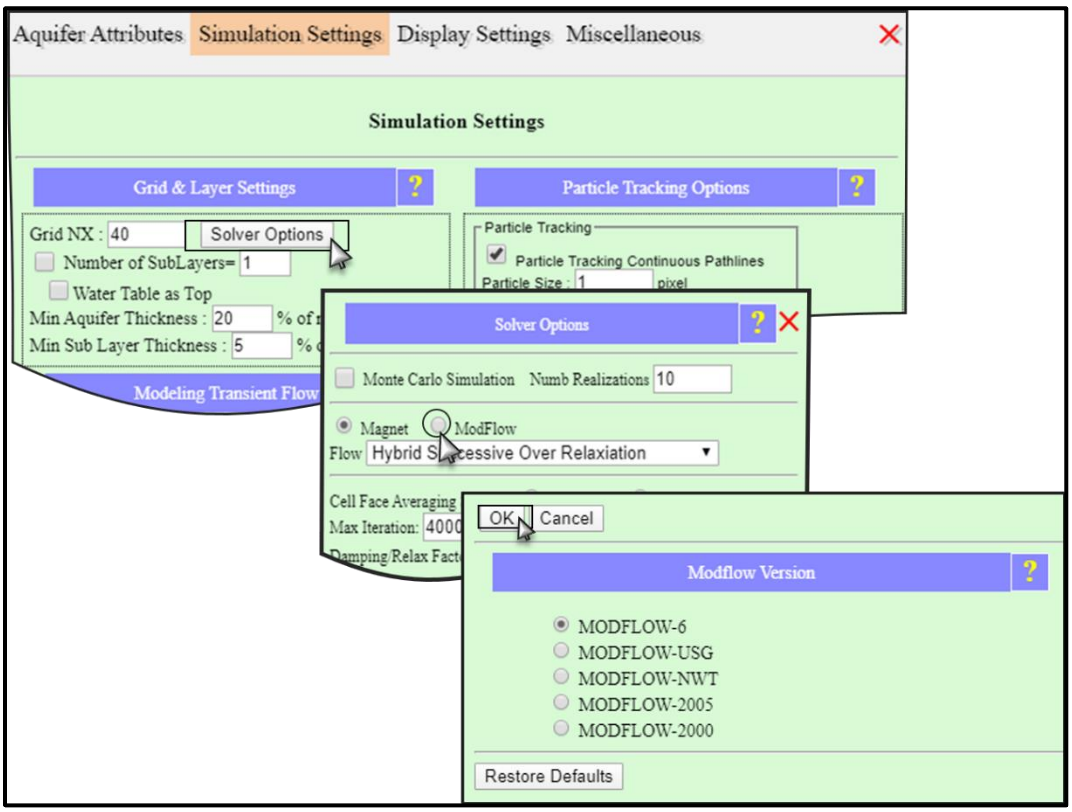


Figure C1: Choosing the MODFLOW engine to solve groundwater flows.

<u>Additional Details</u>:

- By default, the MODFLOW engine uses MODFLOW-6 with settings and parameters sufficient for most models. Different MODFLOW versions, flow formulations, solvers, and additional parameters are available, but not necessary in most cases.