GMS 10.1 Tutorial
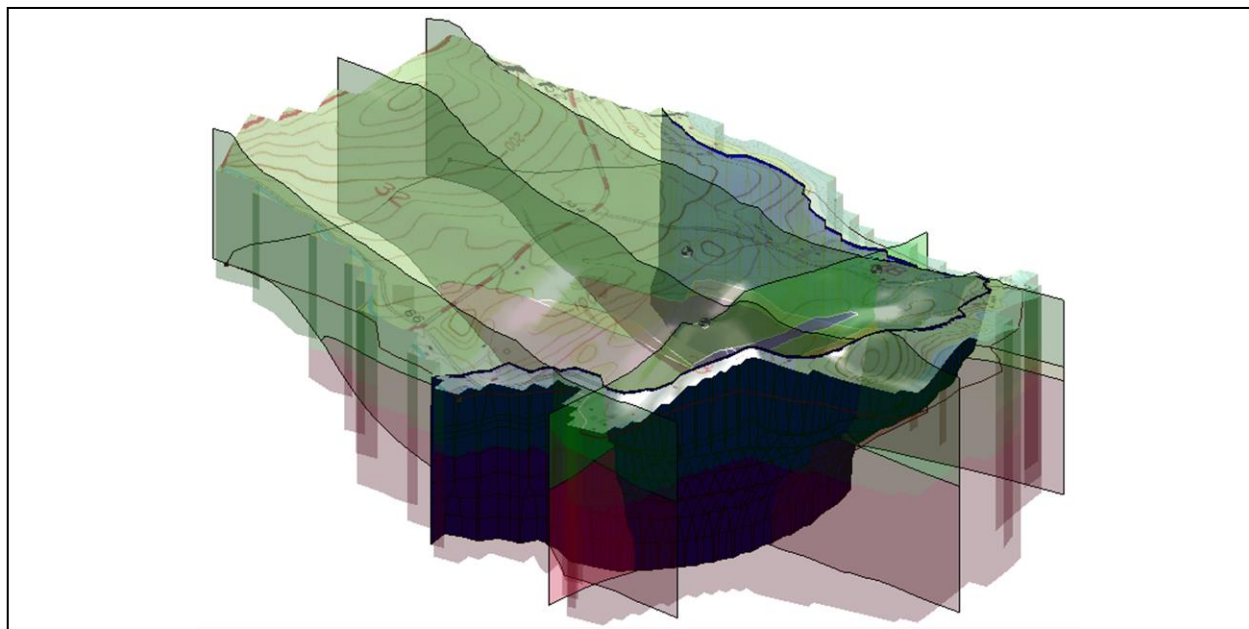
SEAWAT – Conceptual Model Approach

Create a SEAWAT model in GMS using the conceptual model approach

Objectives

Learn to create a SEAWAT model in GMS using the conceptual model approach. Use the GIS tools in the Map module to develop a conceptual model of the site being modeled, including sources/sinks, layer parameters (such as hydraulic conductivity), model boundaries, and all other data necessary for the simulation. Then generate a grid.

Prerequisite Tutorials

- MODFLOW – Conceptual Model Approach
- MT3DMS – Conceptual Model Approach

Required Components

- Grid Module
- Map Module
- MODFLOW
- MT3DMS
- SEAWAT

Time

- 25–40 minutes

AQUAVEO™

1   Introduction

SEAWAT is described on the official website as "a generic MODFLOW/MT3DMS-based computer program designed to simulate three-dimensional variable-density groundwater flow coupled with multi-species solute and heat transport. The program has been used for a wide variety of groundwater studies including those focused on brine migration in continental aquifers as well as those focused on saltwater intrusion in coastal aquifers. SEAWAT uses the familiar structure of MODFLOW and MT3DMS."[1]

Use this model to simulate the effect of the pumping wells on salt water intrusion. The initial heads and concentrations were created by running a SEAWAT model for a very long time with a steady state flow model.

The site in this model is a small coastal aquifer with three production wells with variable pumping rates (Figure 1). The no-flow boundary on the upper left corresponds to a parallel flow boundary, and the no-flow boundary on the left corresponds to a thinning of the aquifer due to a high bedrock elevation. A stream provides a specified head boundary on the lower left, and the remaining boundary is a coastal boundary simulated with a specified head boundary condition.

---

[1] *SEAWAT: A Computer Program for Simulation of Three-Dimensional Variable-Density Ground-Water Flow and Transport*. U.S. Geological Survey. October 7, 2015. http://water.usgs.gov/ogw/seawat/. Accessed July 6, 2017. (Archived by WebCite at http://www.webcitation.org/6ioGQwVBw)
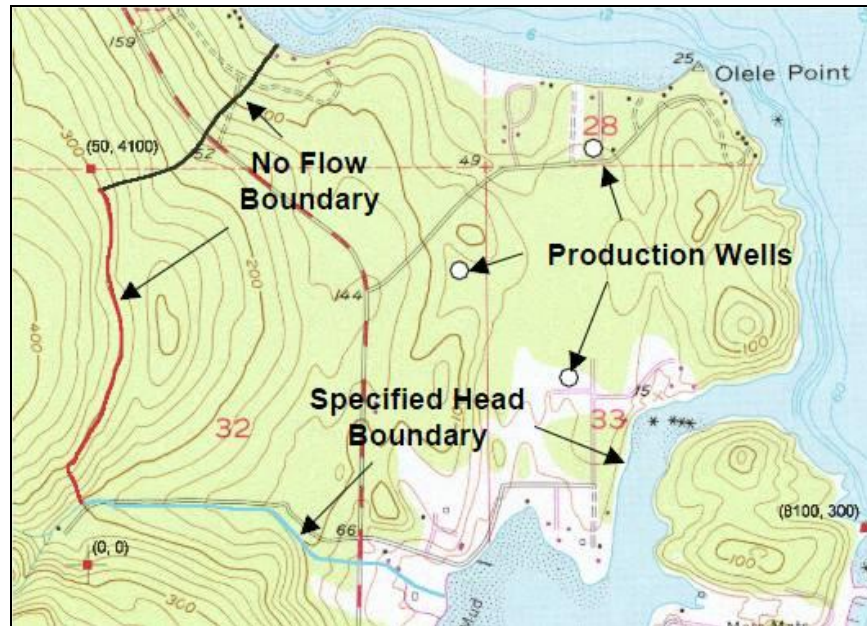
Figure 1   Site map of SEAWAT model

The stratigraphy of the site (Figure 2) consists mainly of an upper and lower aquifer. The upper aquifer has a hydraulic conductivity of 1.5 m/day, and the lower aquifer has a hydraulic conductivity of 5.0 m/day. The model also has some areas with confining units. The production wells extend to the lower aquifer.

Figure 2   Cross section through model domain

This tutorial explains how to perform a SEAWAT simulation within GMS using the conceptual model approach by importing an existing MODFLOW simulation and conceptual model, creating a SEAWAT model to simulate the effects of pumping on salt water intrusion, running the simulation, and viewing the results in 3D.

It is recommended to complete the "MODFLOW – Conceptual Model Approach" and the "MT3DMS – Conceptual Model Approach" tutorials prior to beginning this tutorial.

2   Getting Started

Do the following to get started:

1. If necessary, launch GMS.

2. If GMS is already running, select *File* / **New** to ensure that the program settings are restored to their default state.

2.1   Importing an Existing MODFLOW Model

Start with an existing MODFLOW model and make the necessary modifications to run SEAWAT.

1. Click **Open** 📂 to bring up the *Open* dialog.

2. Select "Project Files (*.gpr)" from the *Files of type* drop-down.

3. Browse to the *\Tutorials\SEAWAT\coastal* directory and select "coastal.gpr".

4. Click **Open** to import the project and exit the *Open* dialog.

The imported model shows a MODFLOW grid on top of a background topo map (Figure 3). The contours on the grid are the heads computed by the long SEAWAT run with a steady state flow model.
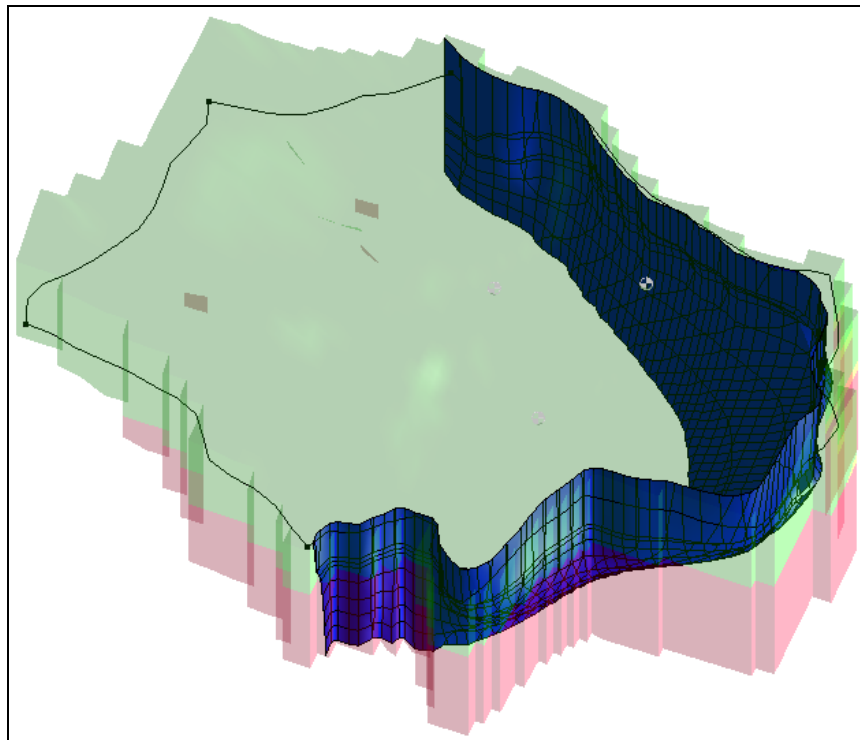

Figure 3   Initial model

2.2   Viewing the Initial Concentration

Now view the initial concentration that will be assigned to the SEAWAT model.

1. Fully expand "🔲 3D Grid Data" folder and select the "🔳 Salt Initial Concentration" dataset under "🔲 grid".

2. Expand the "Display Themes" 🔲 folder and select "🔲 Fresh-water Isosurface".

Cross sections should be visible through the stratigraphy at the site (Figure 4).

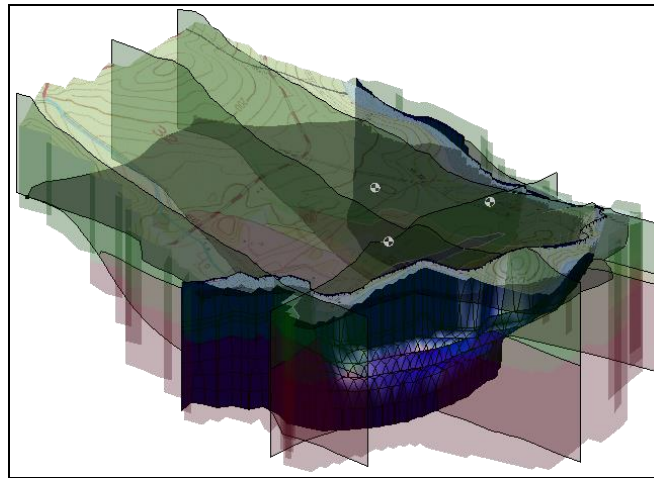3. Use the **Rotate** 🔲 tool to see different angles of the fresh water interface.

Figure 4   Display of the fresh-water isosurface

4. Select "🔲 Isosurface Animation".

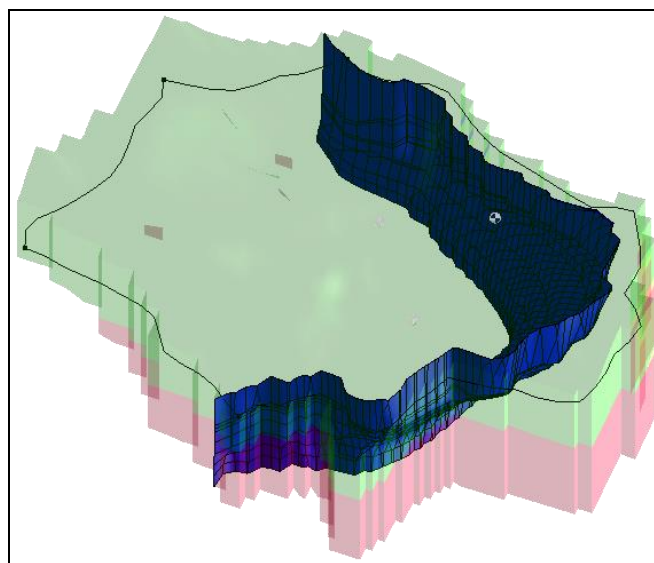This theme only shows the isosurface and the 3D grid faces (Figure 5).

Figure 5   Isosurface and 3D grid faces are visible

5. Select the "Head Contours" ⬚ display theme.

This theme shows the model in **Plan View** ⬚ (Figure 6).
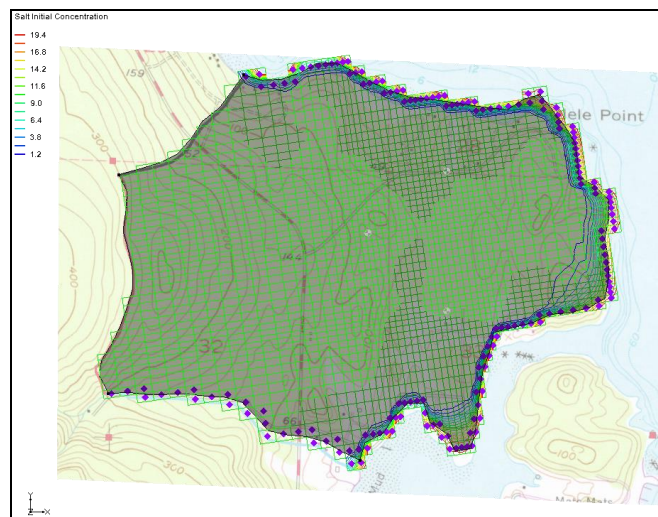

Figure 6   View using Head Contours display theme

2.3   Saving the Project

It is a good idea to save often. Save the project under a new name so that it can be saved periodically.

1. Select *File* / **Save As…** to bring up the *Save As* dialog.

2. Select "Project Files (*.gpr)" from the *Save as type* drop-down.

3. Enter "seawat.gpr" as the *File name*.

4. Click **Save** to save the project under the new name and close the *Save As* dialog.

3   Setting Up the SEAWAT Simulation

Now create a SEAWAT model by modifying the existing MODFLOW model. Do this by editing the conceptual model as well as setting up some options in the MODFLOW, MT3D, and SEAWAT interfaces.

3.1   Editing the MODFLOW simulation

The heads computed by SEAWAT simulation are different than the heads computed by a MODFLOW simulation.[2] The SEAWAT simulation will be transient, so set the starting heads to be those computed by the prior long-term SEAWAT run. It is also necessary to change the MODFLOW model to be transient and to set up the stress periods. The model will run for 730 days, and each stress period will be 10 days long.

1. Select *MODFLOW* | **Global Options…** to open the *MODFLOW Global/Basic Package* dialog.

2. Turn off *Starting heads equal grid top elevation*.

3. Click **Starting Heads…** to open the *Starting Heads* dialog.

4. Click **3D Dataset → Grid…** to open the *Select Dataset* dialog.

5. Select "▦ Starting Heads" in the *Solution* section and click **OK** to close the *Select Dataset* dialog.

6. Click **OK** to exit the *Starting Heads* dialog.

7. In the *Model type* section, select *Transient*.

8. Click **Stress Periods…** to open the *Stress Periods* dialog.

9. Click the **Initialize…** button.

10. Click **Yes** to bring up the *Initialize Stress Periods* dialog when asked to overwrite all current MODFLOW boundary condition data assigned to stress periods.

11. Enter "15" as the *Number of stress periods*.

12. Enter "10.0" as the *Length*.

13. Enter "1" as the *Number of time steps*.

14. Leave the remaining settings at their defaults (Figure 7) and click **OK** to close the *Initialize Stress Periods* dialog.

15. Click **OK** to exit the *Stress Periods* dialog.

16. Click **OK** to exit the *MODFLOW Global/Basic Package* dialog.

---

[2] For more details, see Langevin, Christian. D.; Thorne, Daniel T., Jr.; Dausman, Alyssa M.; Sukop, Michael C.; and Guo, Weixing. (2007). "SEAWAT Version 4: A Computer Program for Simulation of Multi-Species Solute and Heat Transport" in U.S. Geological Survey *Techniques and Methods Book 6*, Chapter A22, p. 13. https://pubs.usgs.gov/tm/tm6a22/pdf/tm6A22.pdf.
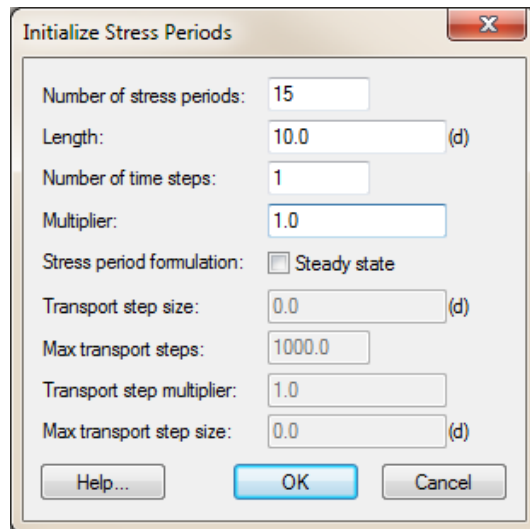
Figure 7   Initialize Stress Periods dialog

3.2   Initializing MT3DMS

As stated earlier, SEAWAT uses a combination of MODFLOW and MT3DMS. Turn on the MT3DMS and SEAWAT menus, then initialize MT3DMS.

1. Right-click on "⊞ grid" and select **New MT3DMS…** to bring up the *Basic Transport Package* dialog.

2. Click **Output Control…** to open the *Output Control* dialog.

3. Enter "40" for the *Print or save at specified interval*.

4. Click **OK** to exit the *Output Control* dialog.

5. Click **Packages…** to open the *MT3DMS/RT3D Packages* dialog.

6. Turn on *Advection package* and *Source/sink mixing package.*

7. Click **OK** to exit the *MT3DMS/RT3D Packages* dialog.

Now define a species and set the starting concentration for the salt species.

8. Click **Define Species…** to open the *Define Species* dialog.

9. Click **New** to create a new species in the spreadsheet on the left.

10. Enter "Salt" in the *Name* column.

11. Click **OK** to exit the *Define Species* dialog.

12. Above the spreadsheet on the right bottom of the dialog, turn on *Use materials for porosity and long. dispersivity*.

13. In the spreadsheet, check the box in the *Edit Per Cell* column of the *Salt* row.

14. Click the ⬚ button in the *Starting Conc. Per Cell* column to open the *Starting Concentrations – Salt* dialog.

15. Click **3D Dataset → Grid** to open the *Select Dataset* dialog.

16. Select "🔳 Salt Initial Concentration" in the *Solution* section and click **OK** to close the *Select Dataset* dialog.

17. Click **OK** to exit the *Starting Concentrations – Salt* dialog.

18. Click **OK** to exit the *Basic Transport Package* dialog.

3.3   Initializing SEAWAT Simulation

With the MODFLOW and MT3DMS models ready, it is possible initialize the SEAWAT simulation

1. Right-click on "⬚ grid" and select **New SEAWAT…** to open the *Global Options* dialog.

2. Turn on *Include transport in simulation*.

3. In the *Packages* section, turn on *Variable-Density Flow (VDF)*.

4. Click **OK** button to exit the *Global Options* dialog.

Modifying the VDF Package

Now enter the data necessary for the VDF package. The inputs to the VDF package control the density calculations performed by SEAWAT. In the example problem, the concentration of seawater was defined as 19 g/l. Based on the units of the MODFLOW model, the reference density will be 1000 kg/m^3. If the density of seawater is 1025 kg/m^3, then the linear relationship between concentration and density is defined with a factor of 1.315.[3]

1. Select *SEAWAT* / **VDF Package…** to open the *SEAWAT VDF Package* dialog.

2. Enter "10.0" for *Length of first transport time step (FIRSTDT)*.

3. Enter "1000.0" for *Reference fluid density (DENSEREF)*.

4. Enter "1.315" for *Density/conc. slope (DRHODC)*.

The dialog should appear as in Figure 8.

5. Click **OK** to exit the *SEAWAT VDF Package* dialog.

---

[3] For more information, see Langevin, et al (2007), pp. 20–21.

Figure 8   SEAWAT VDF Package inputs

3.4   Editing the Conceptual Model

Now adjust the conceptual model by assigning a concentration to the arc defining the coastline. In order to assign concentrations in the conceptual model, turn on transport and define a species.

1.   Fully expand the " Map Data" folder in the Project Explorer.

2.   Right-click on " MODFLOW_SEAWAT" and select **Properties…** to open the *Conceptual Model Properties* dialog.

3.   In the spreadsheet, turn on *Transport*.

4.   Select "MT3DMS" from the *Transport model* drop-down.

5.   Click **Define Species…** to open the *Define Species* dialog.

6.   Click **New** to create a new species in the spreadsheet.

7. Enter "Salt" in the name column and click **OK** to close the *Define Species* dialog.

8. Click **OK** to exit the *Conceptual Model Properties* dialog.

Assigning Salt Concentration at the Coastline

Now that a species has been defined, assign a concentration to the coast line arc.

1. Select " Sources & Sinks" in the Project Explorer to make it active.

2. Using the **Select Arcs** tool, double-click on the coastline arc to bring up the *Attribute Table* dialog.

3. Enter "19.0" in the *Salt conc (g/l)* column.

4. Click **OK** to exit the *Attribute Table* dialog.

Mapping to MT3DMS

Now convert the conceptual model to the numerical model.

1. Select *Feature Objects* / **Map → MT3DMS** to bring up the *Map → Model* dialog.

2. Click **OK** to accept the defaults and close the *Map → Model* dialog.

3. Click **Display Options** to open the *Display Options* dialog.

4. Select "3D Grid Data" from the list on the left.

5. On the *MODFLOW* tab, turn on *Transient head*.

6. Click **OK** to close the *Display Options* dialog.

Notice that all the cells at the coastline now have new symbols representing the boundary condition in the MT3DMS source/sink mixing package (detail in Figure 9).
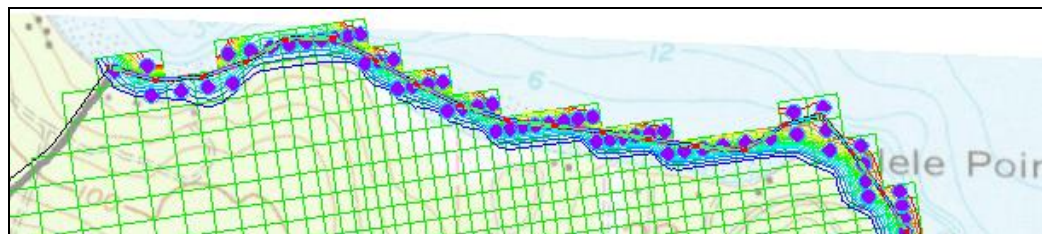

Figure 9   Detail showing the new boundary condition symbols

Mapping to MODFLOW

At this point, it is also necessary to map over the wells and their pumping schedules.

1. Select the " Wells" coverage to make it active.

2. Right-click on "⬤ Wells" and select **Map To → MODFLOW/MODPATH**.

3.5   Saving and Running SEAWAT

Now save the changes and run SEAWAT.

1. **Save** 💾 the project

2. Select *SEAWAT* / **Run SEAWAT…** to bring up the *SEAWAT* model wrapper dialog.

3. When SEAWAT finishes, turn on *Read solution on exit* and *Turn on contours (if not on already)*.

4. Click **Close** to import the solution and close the *SEAWAT* dialog.

4   Viewing the Solution

Now view the results of the SEAWAT model run. Examine how the fresh-water contour line changes with time.

1. Select the "📝 MCL contour" display theme.

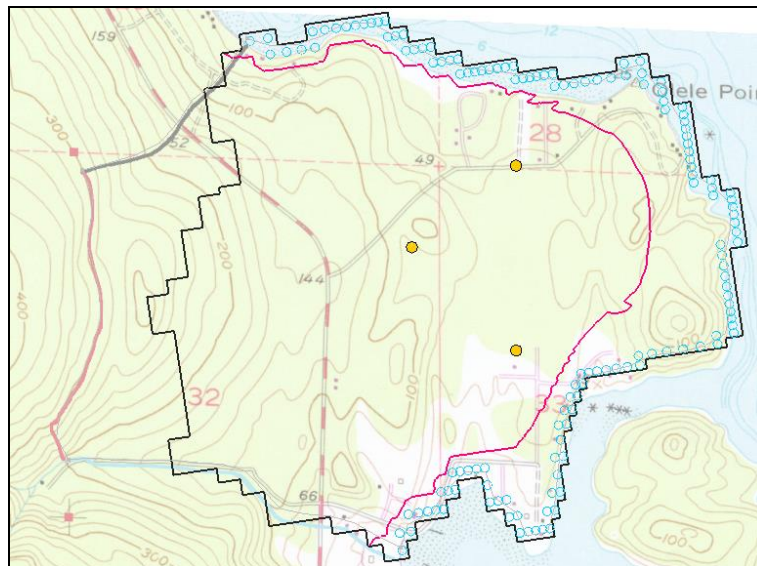This display theme is set up to show one contour at a value of .25 g/l (Figure 10).

Figure 10   Initial appearance after turning on MCL contour display theme

2. Select the "📊 Salt" dataset under the "🔵 seawat (MT3DMS)" solution in the Project Explorer.

3. In the time step window, select time step 12 (it has a value of "120.0").

Notice that the MCL contour moves inland slightly from time step 1 to time step 12 (Figure 11).
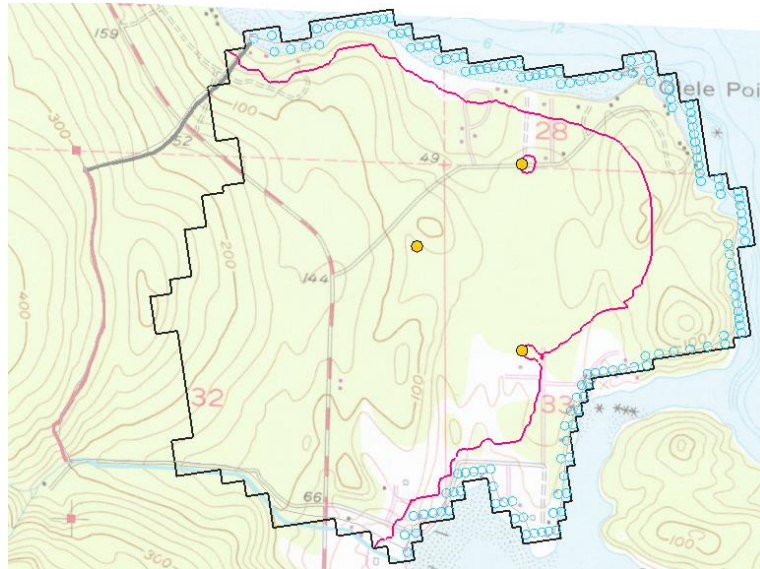

Figure 11   Time step 12

4.1   Creating an Animation

Now create an animation of the change in concentration over time. This helps show when the northernmost well begins pumping lower quality water.

1. Select *Display* / **Animate…** to bring up the *Options* page of the *Animation Wizard* dialog.

The defaults work fine for this animation.

2. Click **Next** to go to the *Datasets* page of the *Animation Wizard* dialog.

3. Click **Finish** to close the *Animation Wizard* dialog.

An animation should begin playing after a few moments (depending on the speed of the computer being used). Notice how the MCL contour line eventually surrounds the northernmost pumping well. Also see how the MCL contour line is moving near the southernmost well.

4. When finished viewing the animation, close the *Play AVI Application* and return to GMS.

4.2   Creating a Time Series Plot of Concentration

Next, make a plot of concentration versus time at the northernmost well.

1. **Zoom** in around the northernmost well.

2. Select "⊞ grid" in the Project Explorer to make it active.

3. Using the **Select Cell** ⊞ tool, select the cell containing the well by clicking on the well.

Notice that the selected cell number is indicated in the status bar at the bottom left of the GMS window ("Cell ID: 13035").

4. Click **Plot Wizard** 📖 to bring up the *Step 1 of 2* page of the *Plot Wizard* dialog.

5. Select "Active Dataset Time Series" from the *Plot Type* list on the left.

6. Click **Finish** to close the *Plot Wizard* dialog.

A plot similar to Figure 12should appear. See how the concentration increases when the well is pumping and then drops off when the well is turned off.
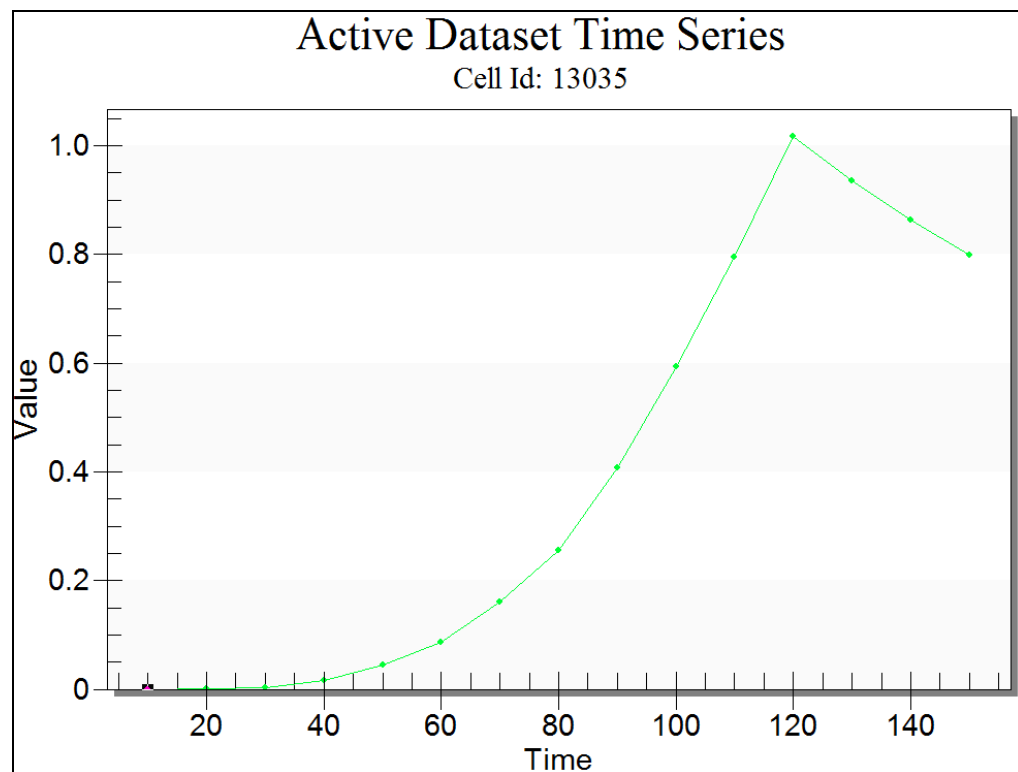

Figure 12   Time series plot of concentration

Close the Plot window and maximize the Graphics Window before continuing.

4.3   Animating the Fresh-Water Surface

Now animate the fresh-water isosurface over time.

1. Select the "Isosurface Animation" 📑 display theme.

2. Select the **Select Material Zones** 📦 tool.

3. Select *Edit* / **Select From List** to bring up the *Select* dialog.

4. Turn on "lower_aquifer" and click **OK** to close the *Select* dialog.

The lower aquifer and the fresh-water isosurface should now be visible. There will also be some circles that are visible. Those can be used to select material zones. To avoid having the circles in the animation, change the tool.

5. Select the **Select Cell** tool.

6. Select *Display* / **Animate…** to bring up the *Options* page of the *Animation Wizard* dialog.

The defaults will work fine for this animation.

7. Click **Next** to go to the *Datasets* page of the *Animation Wizard* dialog.

8. Click **Finish** to close the *Animation Wizard* dialog.

An animation should begin playing after a few moments. Notice how the fresh-water surface is affected by the pumping wells. See how the fresh-water surface passes the northernmost well.

9. Close the *Play AVI Application* to return to GMS.

5   Conclusion

This concludes the "SEAWAT – Conceptual Model Approach" tutorial. The following key concepts were discussed and demonstrated in this tutorial:

- SEAWAT combines MODFLOW and MT3DMS to solve variable density groundwater flow and solute transport problems.

- Use the conceptual model approach with SEAWAT models.

- SEAWAT results can be visualized three dimensionally in GMS.

5.1   Further Reading

—Langevin, Christian D.; Shoemaker, W. Barclay; and Guo, Weixing (2003). *MODFLOW-2000, the U.S. Geological Survey Modular Ground-Water Model– Documentation of the SEAWAT-2000 Version with the Variable-Density Flow Process (VDF) and the Integrated MT3DMS Transport Process (IMT)*: U.S. Geological Survey Open-File Report 03-426, 43 p. http://fl.water.usgs.gov/PDF_files/ofr03_426_langevin.pdf.