GMS 10.4 Tutorial
FEMWATER – Transport Model

Build a FEMWATER model to simulate salinity intrusion

Objectives
This tutorial demonstrates building a FEMWATER transport model using the conceptual model approach. It will review running the model and examining the results.

Prerequisite Tutorials
- FEMWATER – Flow Model

Required Components
- FEMWATER
- Geostatistics
- Map Module
- Mesh Module
- Sub-surface characterization

Time
- 20–30 minutes
1 Introduction

FEMWATER is a three-dimensional finite element groundwater model. It can be used to simulate flow and transport in both the saturated and unsaturated zones. Furthermore, the flow and transport can be coupled to simulate density dependent problems such as salinity intrusion.

The site to be modeled in this tutorial is a small coastal aquifer with three production wells, each pumping at a rate of 2,830 m$^3$/day (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 condition. The coastline arc is assigned with a specified concentration boundary of 19 mg/liter of salt.

The stratigraphy of the site consists of an upper and lower aquifer. The upper aquifer has a hydraulic conductivity of 3 m/day, and the lower aquifer has a hydraulic conductivity of 9 m/day. The wells extend to the lower aquifer. The recharge to the aquifer is about one foot per year. The objective of this tutorial is to create a transport model of the site to simulate salinity intrusion from the coast line.

This tutorial describes how to build a FEMWATER model to simulate salinity intrusion. It will discuss and demonstrate importing an existing FEMWATER flow model, mapping the conceptual model to a FEMWATER simulation, defining additional conditions and running FEMWATER, viewing the water table as an isosurface, and draping the TIFF image on the ground surface.
1.1 Getting Started

Do the following to get started:

1. If GMS is not running, launch GMS.
2. If GMS is already running, select File | New to ensure the program settings are restored to the default state.

2 Opening the Flow Model

Before setting up the FEMWATER transport simulation, there must first be a FEMWATER solution that will be used as the flow field for the transport simulation. In the interest of time, import a previously created FEMWATER simulation.

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 femwater-transport/femwater-transport directory and select “femmod.gpr”.
4. Click Open to import the project file and close the Open dialog.
5. Frame the project.

The Main Graphics Window should appear similar to Figure 2.
3 Building the Transport Model

The purpose of this model is to simulate salinity intrusion by assigning a salt concentration to the coastline arc. The concentration can be assigned directly to the arc in the conceptual model.

3.1 Turning on the Transport Option

1. In the Project Explorer, right-click on “Map Data” and select Expand All.
2. Right-click on “femmod” and select Properties... to open the Conceptual Model Properties dialog.
3. In the table, turn on Transport.
4. Click OK to exit the Conceptual Model Properties dialog.
5. Right-click on “femwater” and select Coverage Setup... to open the Coverage Setup dialog.
6. In the Sources/Sinks/BCs list, turn on Transport BC.
7. Click OK to close the Coverage Setup dialog.

3.2 Defining the Boundary Conditions

Assign boundary conditions to the coastline arc.

1. Select “femwater” to make it active.
2. Using the **Select Arcs** tool, double-click on the coastline arc to bring up the *Attribute Table* dialog.

3. In row 4 in the table, select “spec. conc.” from the drop-down in the *Transport bc* column.

4. Enter “19.0” in the *Conc. (mg/l)* column (scroll to the right, if needed).

5. Click **OK** to close the *Attribute Table* dialog.

### 4 Converting the Conceptual Model

Now it is possible to convert the conceptual model to the 3D mesh model. This will assign all of the boundary conditions using the data defined on the feature objects.

1. In the Project Explorer, right-click on “femmod” and select *Map To* | **FEMWATER** to bring up the *Map → Model* dialog.

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

A set of symbols should appear indicating that the boundary conditions have been assigned (Figure 3).

![Figure 3](image)

*Figure 3* Coastline arc shows the boundary conditions

### 5 Selecting the Analysis Options

Next, select the analysis options.
5.1 Run Options
First, indicate a steady-state flow simulation:

1. Select FEMWATER | Run Options… to open the FEMWATER Run Options dialog.
2. Select “Transport only (1)” for the Type of simulation (OP1).
4. Click OK to close the FEMWATER Run Options dialog.

5.2 Time Control
Second, set the FEMWATER time control options.

1. Select FEMWATER | Time Control… to open the FEMWATER Time Control dialog.
2. Enter “360.0” as the Maximum simulation time.
3. Enter “30.0” as the Constant time step.
4. Click OK to close the FEMWATER Time Control dialog.

5.3 Initial Conditions
Third, set the FEMWATER initial conditions.

1. Select FEMWATER | Initial Conditions… to open the FEMWATER Initial Conditions dialog.
2. In the Files section, click the Open button to the right of Flow (press. head) (FLPH) to bring up the Open dialog.
3. Select “Pressure Head Files (*.phd)” from the Files of type drop-down.
4. Browse to the femwater-transport\femwater-transport\femmod_FEMWATER directory and select “femmod.phd”.
5. Click Open to select the file and exit the Open dialog.
6. In the section on the right, under Flow file format (IVFILE), select Binary (1).
7. Click OK to close the FEMWATER Initial Conditions dialog.

5.4 Output Control
Fourth and finally, have GMS create the concentration dataset solution file.

1. Select FEMWATER | Output Control… to open the FEMWATER Output Control dialog.
2. In the Save options (OC4) section, turn on Save concentration (.con) file (5).
3. Click OK to close the FEMWATER Output Control dialog.
6 Saving and Running the Model

Now to save and run the model.

1. Click Save to save the project file with all of the new settings.

2. Select FEMWATER | Run FEMWATER… to bring up the FEMWATER model wrapper dialog.

The FEMWATER model wrapper dialog should appear showing information on the progress of the model convergence. The model should converge within a few minutes.

3. When the model converges, turn on Read solution on exit and click Close to close the FEMWATER model wrapper dialog.

6.1 Animating the Fresh-water Surface

Now to animate the fresh-water isosurface over time.

1. Right-click on “3D Mesh Data” and select Expand All.

2. Select concentration under “femmod (FEMWATER)”.

3. Select Display | Animate… to bring up the Options page of the Animation Wizard dialog.

4. Click Next to accept the defaults and go to the Datasets page of the Animation Wizard dialog.

5. Click Finish to close the Animation Wizard dialog and create the animation. This process may take a minute or so, depending on the speed of the computer.

The Play AVI Application should open and begin playing the animation. Notice how the freshwater surface is affected by the pumping wells (Figure 4).

6. Once done reviewing the animation, close the Play AVI Application to return to GMS.
7 Conclusion

This concludes the “FEMWATER – Flow Model” tutorial. The following key topics were discussed and demonstrated:

- FEMWATER is a 3D finite element model that is more complex than MODFLOW (which is a 3D finite difference model).
- Setting up a FEMWATER transport model.