

## GMS 10.0 Tutorial

### **FEMWATER – Transport Model**

Build a FEMWATER model to simulate salinity intrusion



#### Objectives

Build a FEMWATER transport model using the conceptual model approach. Run the model and examine the results.

#### Prerequisite Tutorials

- FEMWATER – Flow Model

#### Required Components

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

#### Time

- 20–30 minutes



<b>1</b>	<b>Introduction</b> .....	<b>2</b>
1.1	Outline.....	2
<b>2</b>	<b>Description of Problem</b> .....	<b>2</b>
<b>3</b>	<b>Getting Started</b> .....	<b>3</b>
<b>4</b>	<b>The Flow Model</b> .....	<b>3</b>
<b>5</b>	<b>Building the Transport Model</b> .....	<b>4</b>
5.1	Turning on Transport Option.....	4
5.2	Defining the Boundary Conditions.....	4
<b>6</b>	<b>Converting the Conceptual Model</b> .....	<b>4</b>
<b>7</b>	<b>Selecting the Analysis Options</b> .....	<b>5</b>
7.1	Entering the Run Options.....	5
7.2	Selecting Time Control.....	5
7.3	Selecting Initial Conditions.....	5
7.4	Editing Output Control.....	5
<b>8</b>	<b>Saving and Running the Model</b> .....	<b>6</b>
8.1	Animating the Fresh-water Surface.....	6
<b>9</b>	<b>Conclusion</b> .....	<b>6</b>

# 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 the unsaturated zone. Furthermore, the flow and transport can be coupled to simulate density dependent problems such as salinity intrusion. This tutorial describes how to build a FEMWATER model to simulate salinity intrusion.

## 1.1 Outline

---

Here are the steps to this tutorial:

1. Import an existing FEMWATER flow model.
2. Map the conceptual model to a FEMWATER simulation.
3. Define additional conditions and run FEMWATER.
4. View the water table as an iso-surface.
5. Drape the TIFF image on the ground surface.

# 2 Description of Problem

---

The site to be modeled in this tutorial is shown in Figure 1. The site is a small coastal aquifer with three production wells, each pumping at a rate of 2,830 m<sup>3</sup>/day. 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 specified concentration boundary of 19 mg/l 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.

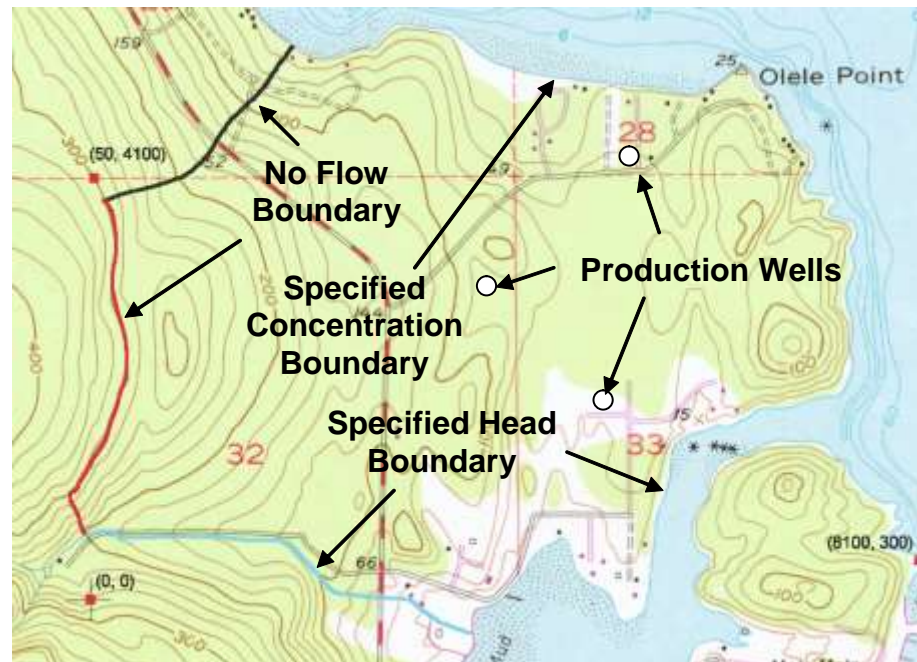


Figure 1. Site to be modeled with FEMWATER.

### 3 Getting Started

---


Do the following to get started:

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

### 4 The Flow Model

---

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

1. Select the **Open**  button.
2. Locate and open the directory entitled `\Tutorials\FEMWATER\femwater-transport`.
3. Select the file entitled “femmod.gpr”.
4. Select the **Open** button.




## 5 Building the Transport Model

---

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

### 5.1 Turning on Transport Option



---

1. In the Project Explorer, expand the “Map Data” folder.
2. In the Project Explorer, right-click on the “femmod”  conceptual model.
3. Select the **Properties** command from the pop-up menu to open the *Conceptual Model Properties* dialog.
4. Turn on *Transport*.
5. Click **OK** to exit the *Conceptual Model Properties* dialog.
6. In the Project Explorer, expand the “femmod”  conceptual model.
7. Right-click the “femwater”  coverage.
8. Select the **Coverage Setup** command from the pop-up menu to open the *Coverage Setup* dialog.
9. In the list of *Sources/Sinks/BCs*, turn on the *Transport BC*.
10. Click **OK**.

### 5.2 Defining the Boundary Conditions

---


Assign boundary conditions to the coastline arc.

1. Select the *Select Arcs*  tool.
2. Click on the coastline arc.
3. Select the **Properties**  button.
4. Change the *Transport bc* to “spec. conc”.
5. Assign the *Conc.* to “19.0” mg/l.
6. Select the **OK** button.

## 6 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 the “femmod”  conceptual model.
2. Select *Map To | FEMWATER* command from the pop-up menu.
3. Select **OK** at the prompt.

A set of symbols should appear indicating that the boundary conditions have been assigned.

## 7 Selecting the Analysis Options

---

Next, select the analysis options.

### 7.1 Entering the Run Options

---

First, indicate a steady-state flow simulation:

1. Select the *FEMWATER* / **Run Options** command to open the *FEMWATER Run Options* dialog.
2. For the *Type of simulation (OPI)*, ensure the “Transport only (1)” option is selected.
3. For *Quadrature (IQUAR)*, select the “Nodal/Nodal” option.
4. Select the **OK** button.

### 7.2 Selecting Time Control

---


Second, select the FEMWATER time control options.

1. Select the *FEMWATER* / **Time Control** command to open the *FEMWATER Time Control* dialog.
2. Set the *Maximum simulation time* to “360”.
3. Set the *Constant time step* to “30”.
4. Select the **OK** button.

### 7.3 Selecting Initial Conditions

---

Next, set the FEMWATER initial conditions.

1. Select the *FEMWATER* / **Initial Conditions** command to open the *FEMWATER Initial Conditions* dialog.
2. For the *Flow (press. head) (FLPH)*, click on **Open** .
3. Browse to `\Tutorials\FEMWATER\femwater-transport\femmod_FEMWATER`
4. Select the existing solution called “femmod.phd”.
5. Select the **Open** button.
6. In the *Flow file format (IVFILE)*, select *Binary (1)*.
7. Select the **OK** button.

### 7.4 Editing Output Control

---


Finally, have GMS write out the concentration dataset solution.

1. Select the *FEMWATER* / **Output Control** command to open the *FEMWATER Output Control* dialog.
2. Turn on *Save concentration (.con) file (5)*.
3. Select the **OK** button.

## 8 Saving and Running the Model

---

Now save and run the model.

1. Select the **Save**  button.
2. Select the *FEMWATER* / **Run FEMWATER** command.




The FEMWATER window should appear and you will see some information on the progress of the model convergence. The model should converge in a few minutes.

3. When the model converges, select the **Close** button.

### 8.1 Animating the Fresh-water Surface

---

Now animate the fresh-water isosurface over time.

1. Select the “Isosurface Animation”  display theme.
2. Select the “concentration”  dataset below the “femmod (FEMWATER)”  solution in the Project Explorer.
3. Select the *Display* / **Animate** command.

The defaults will work fine for this animation.

4. Select **Next** on the first page.
5. Select **Finish** on the second page.

After a minute or so, an animation should begin playing. Notice how the freshwater surface is affected by the pumping wells.

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

## 9 Conclusion

---

This concludes the tutorial. Here are the things that you should have learned in this tutorial:

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