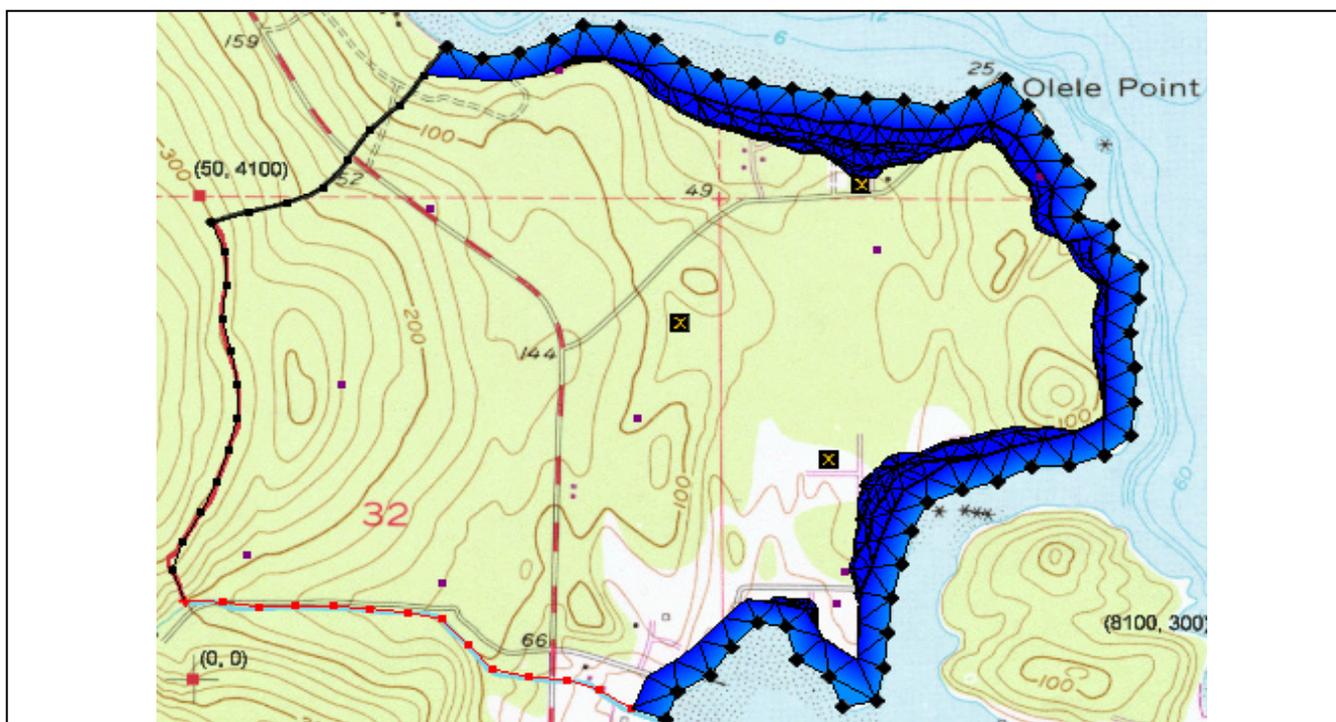


## GMS 9.2 Tutorial

# 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
- Mesh
- Sub-surface characterization

### Time

- 30-60 minutes

**AQUAVEO™**



# 1 Contents

---

<b>1</b>	<b>Contents .....</b>	<b>2</b>
<b>2</b>	<b>Introduction .....</b>	<b>2</b>
2.1	Outline .....	2
<b>3</b>	<b>Description of Problem .....</b>	<b>2</b>
<b>4</b>	<b>Getting Started .....</b>	<b>3</b>
<b>5</b>	<b>The Flow Model.....</b>	<b>3</b>
<b>6</b>	<b>Building the Transport Model .....</b>	<b>4</b>
6.1	Turning on Transport Option.....	4
6.2	Defining the Boundary Conditions .....	4
<b>7</b>	<b>Converting the Conceptual Model.....</b>	<b>4</b>
<b>8</b>	<b>Selecting the Analysis Options .....</b>	<b>4</b>
8.1	Entering the Run Options .....	5
8.2	Selecting Time Control.....	5
8.3	Selecting Initial Conditions .....	5
8.4	Editing Output Control .....	5
<b>9</b>	<b>Saving and Running the Model.....</b>	<b>5</b>
9.1	Animating the Fresh-water Surface .....	6
<b>10</b>	<b>Conclusion.....</b>	<b>6</b>

## 2 Introduction

---

FEMWATER is a three-dimensional finite element ground water 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.

### 2.1 Outline

---

This is what you will do:

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.

## 3 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.

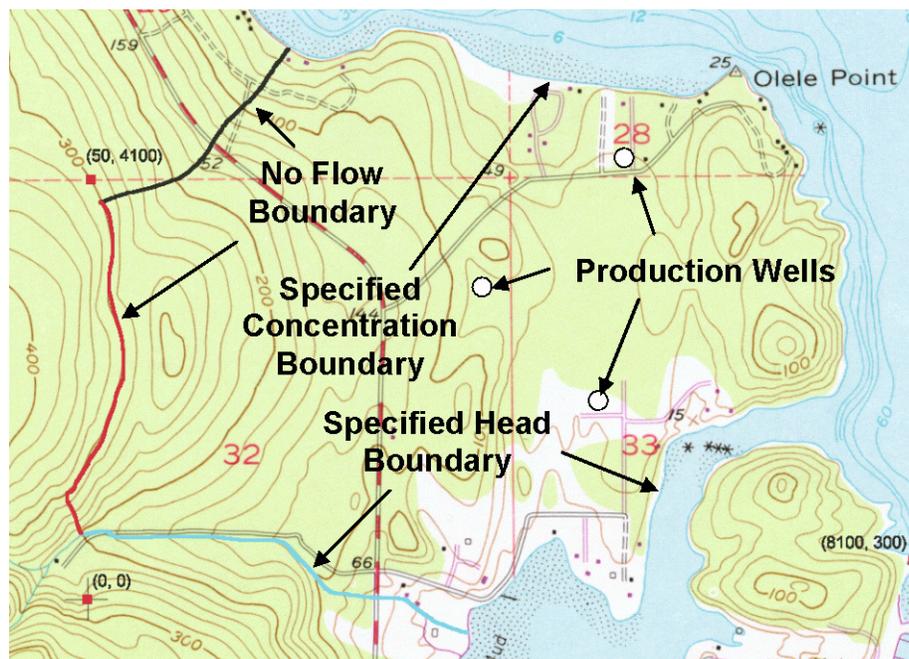


Figure 1. Site to be modeled with FEMWATER.

## 4 Getting Started

If you have not yet done so, launch GMS. If you have already been using GMS, you may wish to select the *File | New* command to ensure the program settings are restored to the default state.

## 5 The Flow Model

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

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

## 6 Building the Transport Model

---

The purpose of our model is to simulate salinity intrusion. We will assign a salt concentration to the coastline arc. The concentration can be assigned directly to the arc in the conceptual model.

### 6.1 Turning on Transport Option

---

1. In the *Project Explorer*, right-click on the **femmod** conceptual model  and select the *Properties* command from the pop-up menu.
2. Turn **on** *Transport*.
3. Click *OK* to exit the *Conceptual Model Properties* dialog.
4. In the *Project Explorer*, right click the **femwater** coverage  and select the *Coverage Setup* command from the pop-up menu.
5. In the list of *Sources/Sinks/BCs*, turn **on** the *Transport BC*.
6. Click *OK*.

### 6.2 Defining the Boundary Conditions

---

We will 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 concentration to **19.0** mg/l.
6. Select the *OK* button.

## 7 Converting the Conceptual Model

---

Now we are ready 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  and select *Map* → *FEMWATER* command from the pop-up menu.

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

## 8 Selecting the Analysis Options

---

Next, we will select the analysis options.

---

## 8.1 Entering the Run Options

---

First, we will indicate that we wish to run a steady state flow simulation:

1. Select the *FEMWATER | Run Options* command.
2. For the *Type of simulation (OPI)*, ensure the **Transient only (1)** option is selected.
3. In the *Quadrature selection* section, select the **Nodal/Nodal** option.
4. Select the *OK* button.

---

## 8.2 Selecting Time Control

---

Second, we will select the FEMWATER time control options.

1. Select the *FEMWATER | Time Control* command.
2. Set the *Maximum simulation time* to **360**.
3. Set the *Constant time step* to **30**.
4. Select the *OK* button.

---

## 8.3 Selecting Initial Conditions

---

Next, we will set the FEMWATER initial conditions.

1. Select the *FEMWATER | Initial Conditions* command.
2. For the *Flow (press. head) (FLPH)*, click on .
3. Browse to the existing solution called *femmod.phd*.
4. Select the *OK* button.
5. In the *Flow file format (IVFILE)*, select *Binary (1)*.
6. Select the *OK* button.

---

## 8.4 Editing Output Control

---

Final, we need to have GMS written the concentration data set solution out.

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

---

## 9 Saving and Running the Model

---

We are now ready to save and run the model.

1. Select the *Save* button .

To run FEMWATER:

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.

## 9.1 Animating the Fresh-water Surface

---

Now we will animate the fresh-water isosurface over time.

1. Select the **Isosurface Animation**  display theme.
2. Select the **concentration**  data set 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 and *Finish* on the second page.

After a few minutes an animation should begin playing. Notice how the fresh-water surface is affected by the pumping wells.

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

## 10 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.