SEAWAT Tutorial - Elder Model

Introduction

Saltwater intrusion is a common occurrence along coastlines throughout the world. Saltwater is denser than freshwater, and consequently will tend to migrate inland underlying freshwater zones when groundwater resources are depleted or in areas where the water table is low. A static equilibrium is reached when the freshwater discharge force balances the buoyancy force due to density. Visual MODFLOW includes the SEAWAT Engine to simulate saltwater intrusion.

About SEAWAT

SEAWAT a computer program for simulation of three-dimensional, variable-density, transient ground-water flow in porous media. SEAWAT was designed by combining a modified version of MODFLOW-2000 and MT3DMS into a single computer program. SEAWAT contains all of the processes distributed with MODFLOW-2000 and also includes the Variable-Density Flow Process (as an alternative to the constant-density Ground-Water Flow Process) and the Integrated MT3DMS Transport Process.

Background

This tutorial illustrates a common application for SEAWAT and is based on the Elder Problem published in User's Guide to SEAWAT.

Elder model is commonly used as a benchmark problem for variable-density ground-water flow, where fluid density is a function of salt concentration. This problems simulates salt-water intrusion and proliferation through a freshwater aquifer. Molecular diffusion is the sole mechanism of hydrodynamic dispersion during the simulation, which runs for 20 years.Salt from the constant-concentration boundary at the top of the model diffuses into the model domain and initiates series of complex vortices that redistribute the salt mass throughout the model. A constant-concentration boundary with a value of zero is specified for the lowest layer in the model. Two outlet cells with constant head values of zero are specified for the upper left and right boundaries. These constant head cells allow salt to diffuse into the model by providing an outlet for fluid and salt mass (User's Guide to SEAWAT: A Computer Program For Simulation of Three-Dimensional Variable-Density Grown-Water Flow. U.S. Geological Survey Open-File Report 01-232).

Terms and Notations

For the purposes of this tutorial, the following terms and notations will be used:

Type: - type in the given word or value

⇔ - press the Tab key on your keyboard

<Enter> - press the Enter key on your keyboard

☞ - click the left mouse button where indicated

☞☞ - double-click the left mouse button where indicated

The **bold faced type** indicates menu or window items to click on or values to type in.

[...] - denotes a button to click on, either in a window, or in the side or bottom menu bars.

Creating the Visual MODFLOW Model

In the first portion of the tutorial you will build the model for the simulation.

NOTE: Some features described in this tutorial are only available in a Pro or Premium version.

On your Windows desktop, you will see an icon for Visual MODFLOW.

☞☞ Visual MODFLOW to start the program

To create your new model:

☞ **File** from the top menu bar

☞ **New**

A **Create new model** window will appear.

For your convenience, a model following this step-by-step tutorial has already been created and is included with the installation. The model is located in the default directory **C:\My Documents\Visual MODFLOW\Tutorial\SEAWAT**.

To prevent overwriting these files, it is recommended that you create a new folder, and save the new model in this folder.

Create a new folder.

Type in a new folder name.

☞☞ on the new folder

Type: **SEAWAT** in the **File name** field

☞ **[Save]**

NOTE: If you choose the default directory (**C:\My Documents\Visual MODFLOW\Tutorial\SEAWAT**) and type in the same **SEAWAT** file name, a warning message will appear.

- ☞ **[No]** to save your new model in a different folder, or
- ☞ **[Yes]** to overwrite the existing model.

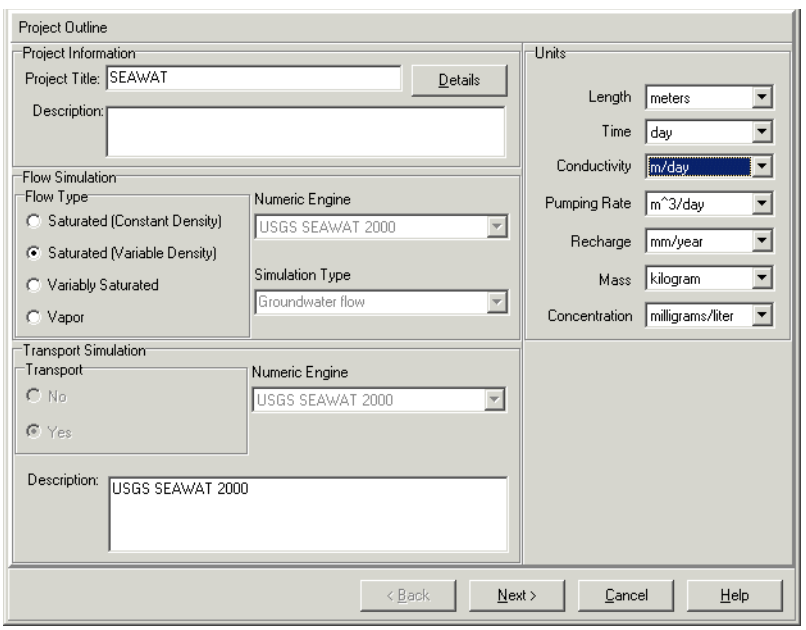
Visual MODFLOW automatically adds a .**VMF** extension onto the end of the filename.

Next, the Model Setup is described in four consecutive steps.

Step 1

In Step 1, the following window is used to define:

- the desired **Project Information**
- **Units** associated with various flow and transport parameters
- **Flow Type** and **Transport Simulation**, which will narrow the choice for the desired **Engines**



Project Information frame

The project name (SEAWAT) is displayed in the **Project Title** field. If you wish, you can enter project details (e.g. “This project represents a modified version of the Elder problem”), or click the **[Details]** button to enter more detailed information about the project.

Flow Simulation frame

☞ **Saturated (Variable Density)** flow type radio button in the **Flow Type** frame.

USGS SEAWAT is automatically selected as the **Numeric Engine** and Groundwater flow as the **Simulation Type**. As these are the only options available for the **Saturated (Variable Density)** flow type, you do not have the ability to select any other numeric engine and/or simulation type.

Transport Simulation frame

The settings in this frame are automatically set for the USGS SEAWAT .

Units frame

Set the following units:

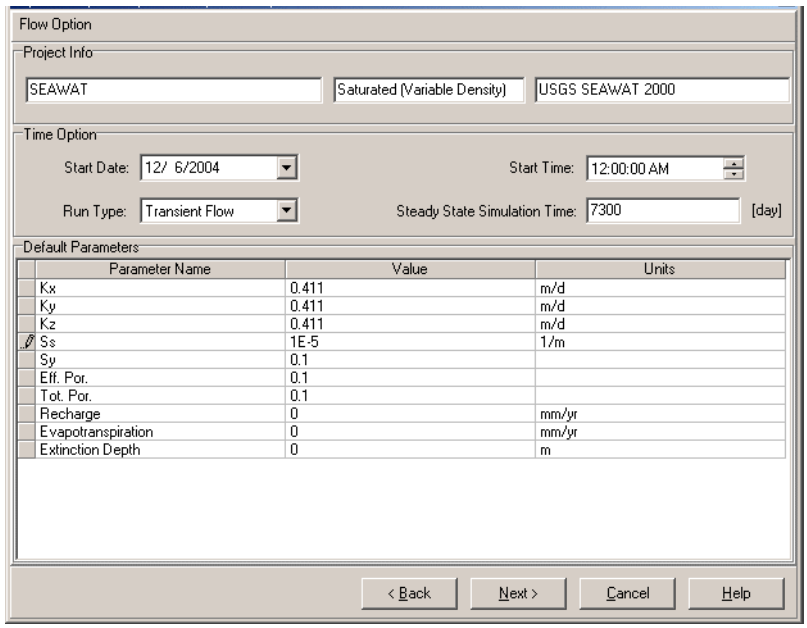
- Length: meters
- Time: day
- Conductivity: m/day
- Pumping rate: m^3/day
- Recharge: mm/year
- Mass: kilogram
- Concentration: milligrams/liter

When you are finished entering the information pertinent to this step, the dialogue should appear similar to the one shown above.

☞ **[Next >]** to move to the next step

Step 2

In Step 2, the following window will appear displaying the default parameter values for the Flow simulation.



Project Info frame

The project information frame now includes the name of the project, the flow type, and the numeric engine used in the project.

Time Option frame

The **Start Date** and **Start Time** of the model correspond to the beginning of the simulation time period. Currently, this date is relevant only for transient flow simulations where recorded field data may be imported for defining time schedules for selected boundary conditions (Constant Head, River, General Head and Drain).

You will simulate processes over 20 years, however the time in the model is counted in days, which makes it 7300 days.

Type: 7300 for the Steady State Simulation time

Default Parameters frame

This frame allows you to enter some flow parameters that will be considered default for your model. Typical parameter values are already entered, however you will change some of them:

- For **Kx**, **Ky**, and **Kz**

Type: 0.411

- For **Sy**, **Effective Porosity**, and **Total Porosity**

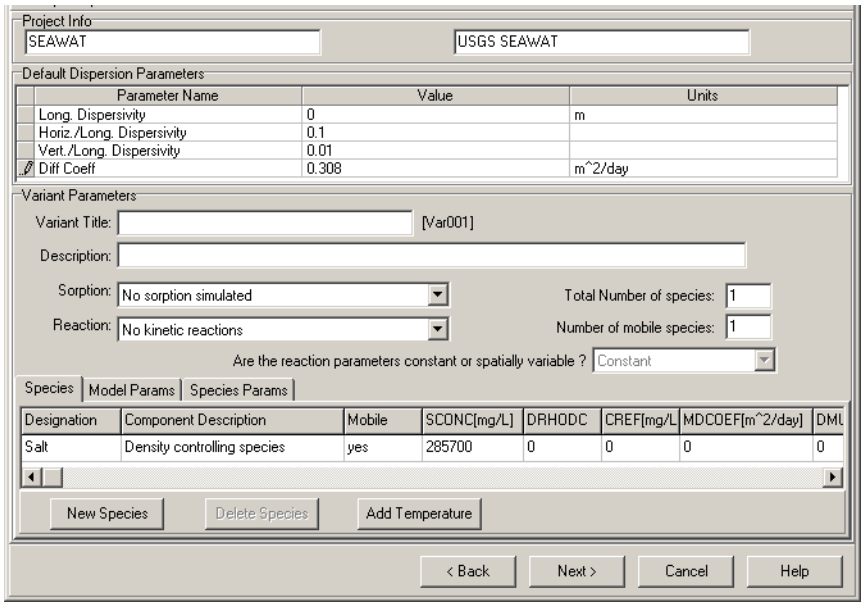
Type: 0.10

Leave the rest of the values as default. Once you are finished entering the parameter values, the dialogue should look similar to the one shown above.

☞ **[Next >]** to move on to the next step

Step 3

In Step 3, the following window will appear displaying the default parameter values for the Transport simulation.



Project Info frame

This frame contains the project name and the numeric engine used for the project.

Default Dispersion Parameters frame

This frame allows you to set the dispersion parameters that will be used in the model. These parameters can be modified later.

- For **Long. Dispersivity**

Type: 0

- For **Diff Coeff**

Type: 0.308

Leave the rest of the values as default.

Variant Parameters frame

In order to support all of the available options for the multi-species reactive transport programs, Visual MODFLOW requires you to setup the initial conditions for the contaminant transport scenario (e.g. number of chemical species, names of each chemical species, initial concentrations, decay rates, partitioning coefficients, etc.). Each scenario is referred to as a Transport Variant, and you can have more than one variant for a given flow model.

By default Visual MODFLOW creates a variant named **VAR001**.

We will now edit this variant to setup the model for transport processes.

For this model, there is no sorption or kinetic reactions.

At the bottom of the dialogue there are three tabs that allow you to set up the variables.

- **Species**
- **Model Params**
- **Species Params**

In the **Species** tab you will see a spreadsheet view with labeled column headings. The chemical species being simulated in this model is Salt. Visual MODFLOW allows you to use the real names of the simulated chemicals in order to easily identify them in contour maps and graphs.

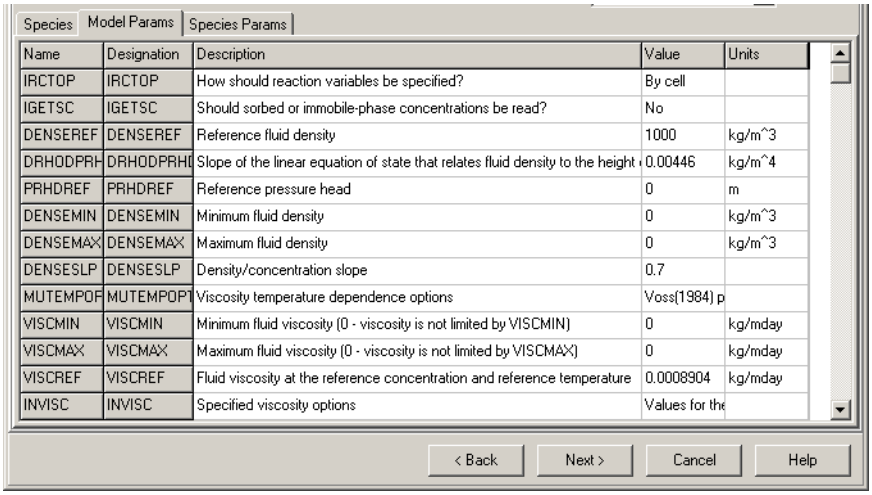
Salt is created as the default variable when you select saturated (variable density) flow type.

In the **SCONC** ((mg/L))

Type: 285700

This is equivalent to the 285.7 kg/m^3 concentration. After this step is complete, the dialogue should be similar to the one shown above.

☞ **Model Params** tab



Set:

- For **Minimum fluid density**

Type: 0

- For **Maximum fluid density**

Type: 0

- For **Density/concentration slope**

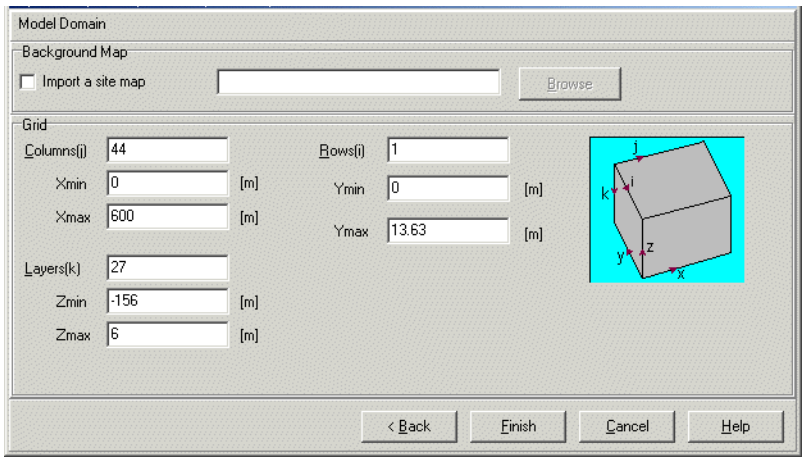
Type: 0.7

Leave the rest of the values as default.

☞ **[Next >]** to move to the next step

Step 4

Step 4 is creating the model grid (see figure below).



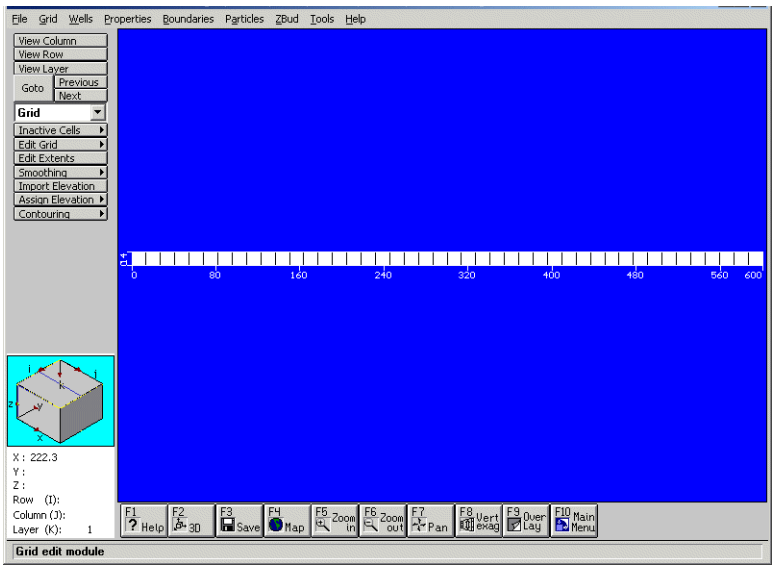
The above window is used to **Import a site map**, specify the dimensions of the **Model Domain**, and define the number of rows, columns, and layers for the finite difference grid.

Enter the following number of rows, columns, and layers to be used in the model. **Type** the following into the **Grid** fields,

Columns (j):	**44**
X min:	**0**
X max:	**600**
Rows (i):	**1**
Y min:	**0**
Y max:	**13.63**
Layers (k):	**27**
Zmin:	**-156**
Zmax:	**6**

☞ **[Finish]** to complete the project set-up

The model setup wizard is now completed, and the **Input** menu will open with an automatically generated uniformly spaced 44 x 1 x 27 finite-difference grid. The Grid will appear on the screen, as shown in the following figure:

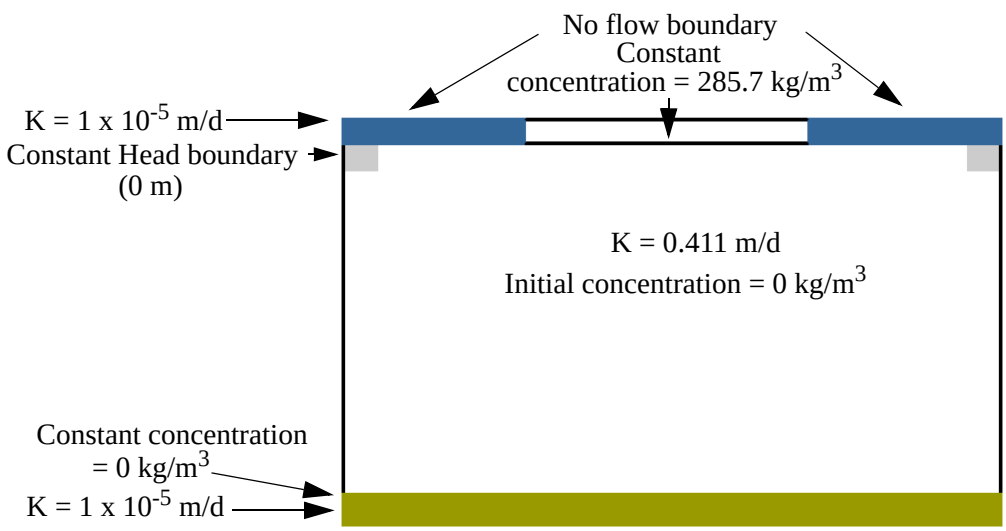


By default the **Grid** screen is loaded when you first enter the **Input** module.

Input

To simulate salt water infiltration into a freshwater aquifer, you need to assign various properties and boundaries to your model.

Below is a schematic (cross-sectional) representation of these properties and boundaries:



First, you will define the boundary conditions.

☞ **View Row** from the side menu

Move the cursor over the model and it will highlight in pink.

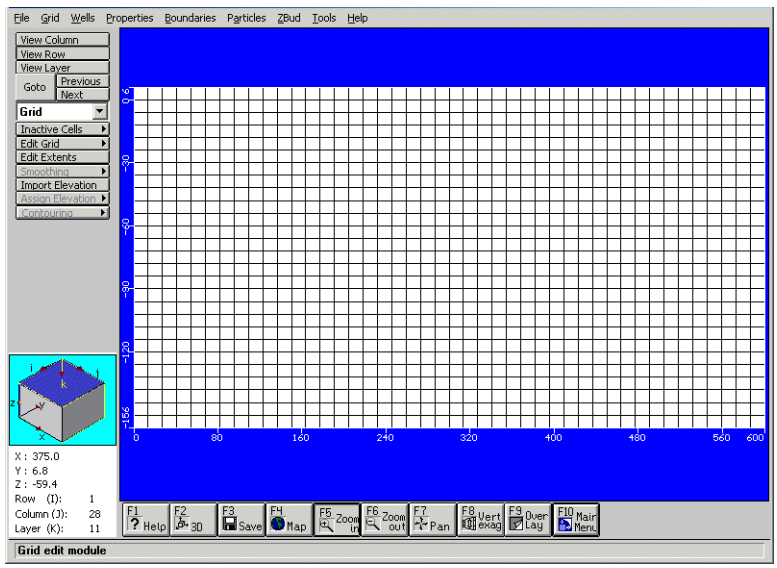
☞ The highlighted row

☞ **F8 Vert Exag** button below the model, or press the **F8** button on your keyboard

Type: 2 as the **Vertical Exaggeration**

☞ **[OK]** to apply the change

The input window should now resemble the following figure:

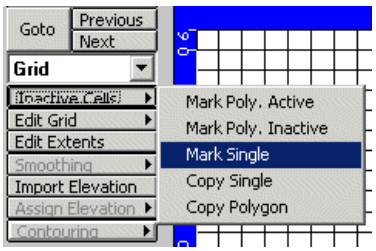


The lower left corner of the window contains the navigation cube as well as X, Y, and Z coordinates and the Row, Column, and Layer which is currently under the tip of the

cursor. These tools are invaluable in precisely positioning various model attributes; refer to them, while following the instructions below.

No-flow boundary

☞ **Inactive Cells>Mark Single** from the side menu



☞ In the Row1-Column1-Layer1 cell (top-left corner cell of the model)

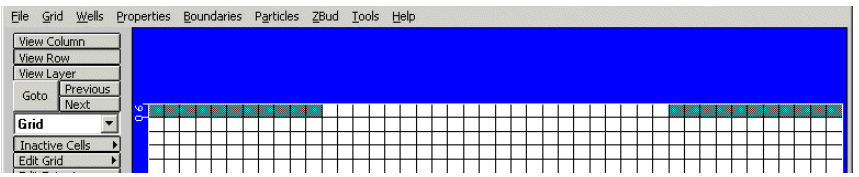
The cell will change color to green.

☞ In the Row1-Column2-Layer1 cell

This cell will also change color to green. Continue across the model to mark 11 cells in the first layer (i.e. the last cell you mark should be Row1-Column11-Layer1).

Move to the right side of the model and mark the 11 cells on the opposite end (i.e. cells Row1-Column34-Layer1 through Row1-Column44-Layer1)

The top of your model should now appear as shown below:



Conductivity

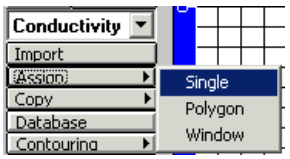
There are two zones of hydraulic conductivity in this model. Large portion of the model domain (Layer 2 to 26) uses the default conductivity of 0.411 m/day, however the top and the bottom layer (Layer 1 and Layer 27) have conductivity of 1.0×10^{-5} m/day.

To assign the appropriate conductivity

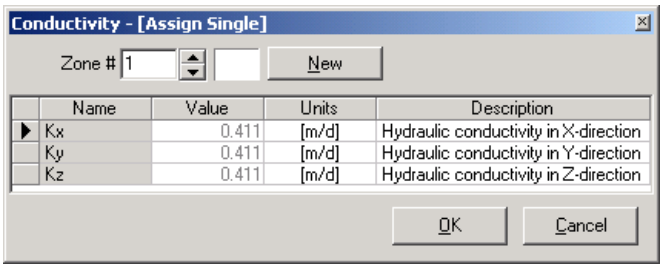
☞ **Properties>Conductivity** from the menu

☞ **[Yes]** to save the changes

☞ **Assign>Single** from the side menu



The following dialogue will load:



Zone #1 is the default conductivity zone and is created over the entire model according to the values specified when the project was first created. To define a new conductivity zone,

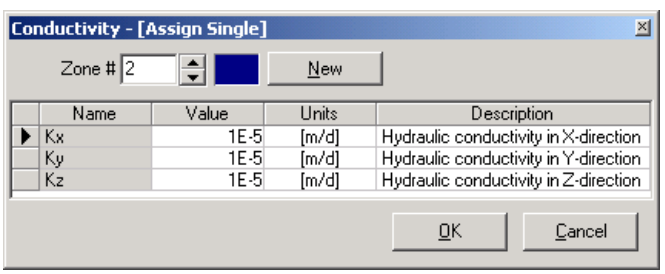
☞ **[New]**

Note that the color of the box between the **Zone #** field and the **[New]** button has changed. The cells that have the new conductivity will be shown in this color.

- For **Kx**, **Ky**, and **Kz**

Type: 1E-5

The dialogue should now appear as follows:



☞ At Row1-Column1-Layer1 (and hold down the left mouse button)

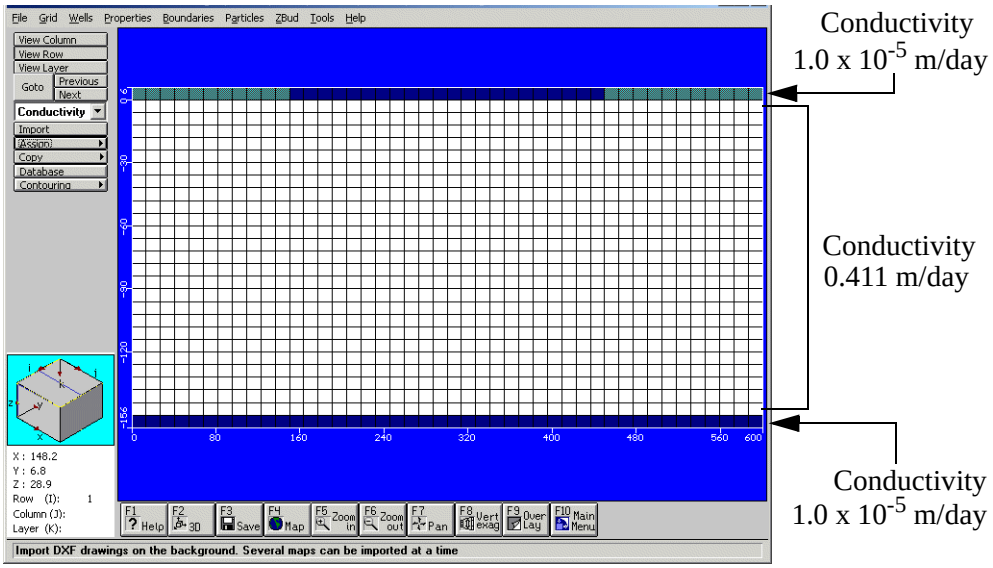
Without releasing the left mouse button, mark the entire top layer (Layer1) with the new Zone. Then release the mouse and in the same manner mark the bottom layer (Layer27). The cells will change color as you assign them.

If you made a mistake and marked the wrong cell, you can return it to Zone 1 by clicking the to chose Zone 1 and then selecting the mis-assigned cell.

When you are finished,

☞ **[OK]** to save the changes and close the dialogue

The model should now appear as shown below:



Constant Concentration boundaries

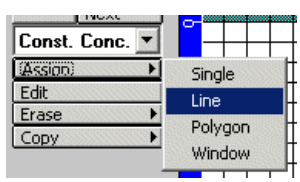
This model will simulate the intrusion of salt water into the freshwater aquifer. As such, there is a boundary with the constant salt concentration of 285.7 kg/m^3 at the top of the model (Layer1) and a boundary with the constant concentration of 0 kg/m^3 at the bottom of the model (Layer27).

To assign these boundaries

☞ **Boundaries>Constant Concentration**

☞ **[Yes]** to save the changes

☞ **Assign>Line** from the side menu



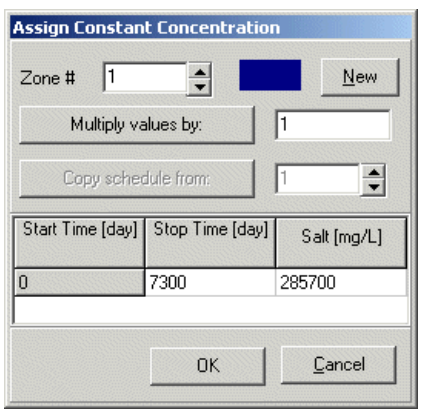
You will assign the constant concentration of 285.7 kg/m^3 to the portion of Layer1 between the inactive cells:

- ☞ In the Row1-Column12-Layer1 cell to establish the beginning of the line
- ☞ Right-click in Row1-Column33-Layer1 cell to end the line

The selected cells will change color to Zone 1 color, and the **Assign Constant Concentration** dialogue will load.

Type: 7300 in **Stop Time (day)** field

Type: 285700 in **Salt (mg/l)** field



☞ **[OK]** to save the changes and close the dialogue

Now you need to assign a boundary with constant concentration of 0 kg/m^3 to Layer27.

☞ **Assign>Line** from the side menu

☞ In the bottom left corner cell (Row1-Col1-Layer27)

☞ Right-click in the bottom right corner cell (Row1-Col44-Layer27)

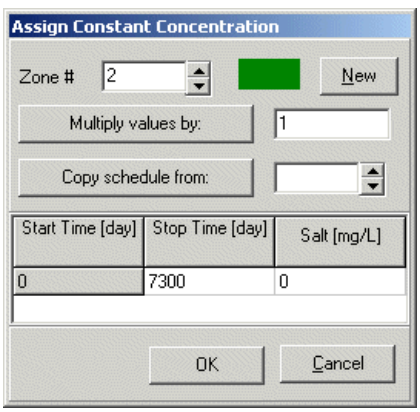
When the **Assign Constant Concentration** dialogue loads,

☞ **[New]** to create a new concentration zone

The color of the selected cells, as well as the color of the zone in the dialogue will change to green.

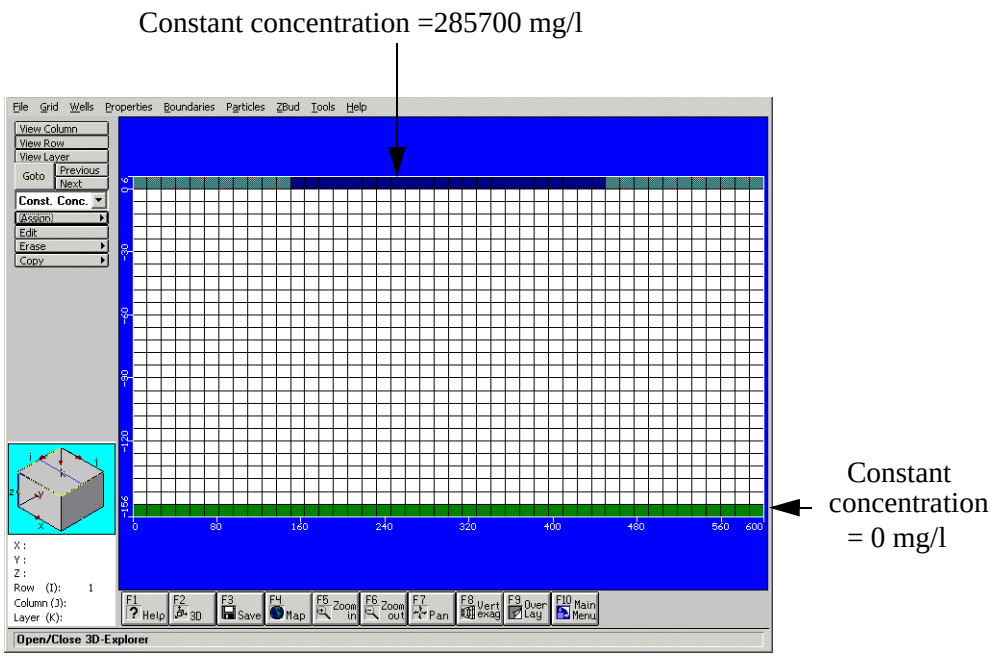
Type: 7300 in **Stop Time (day)**

Type: 0 in **Salt (mg/l)**



☞ **[OK]** to save the changes and close the dialogue

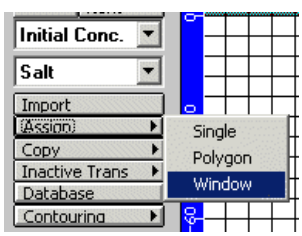
Your model should now appear as follows:



Initial Concentration

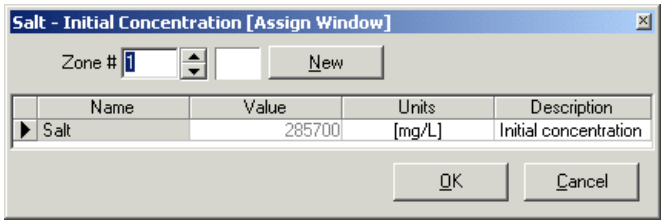
Between the two constant concentration boundaries is a freshwater aquifer with an initial salt concentration of 0 kg/m^3. When the model was first created, the default initial concentration of 285.7 kg/m3 was assigned to the entire model domain. To assign this new property,

- ☞ **Properties>Initial Concentration**
- ☞ **[Yes]** to save the changes
- ☞ **Assign>Window** from the side menu



Mark a rectangular area that spans from Row1-Column1-Layer2 cell to Row1-Column44-Layer27 cell (entire model excluding the first layer) by clicking once in each of the cells specified above.

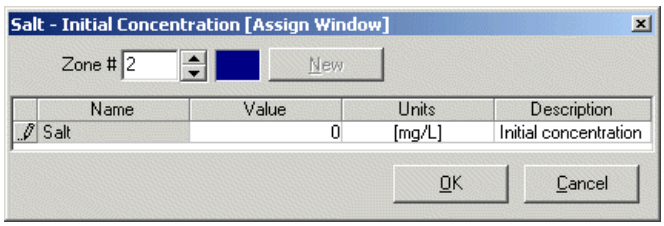
Once the window is created, the following dialogue will be loaded:



☞ **[New]** to define a new initial concentration zone for the region you just defined

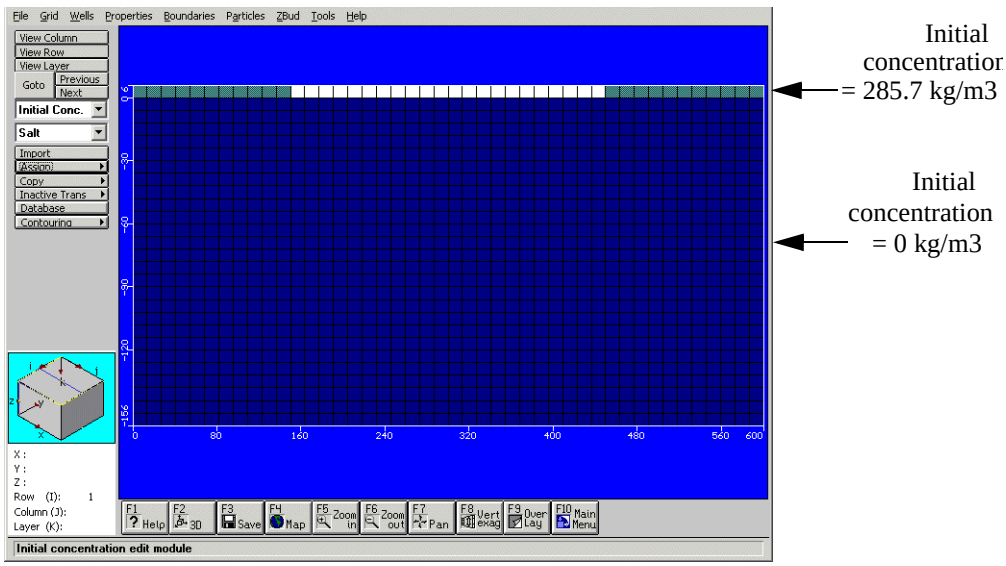
The selected cells will change color to match the new zone

Type: 0 in **Value** column for the Salt concentration



☞ **[OK]** to save the changes and close the dialogue

Your model should now look similar to the following:

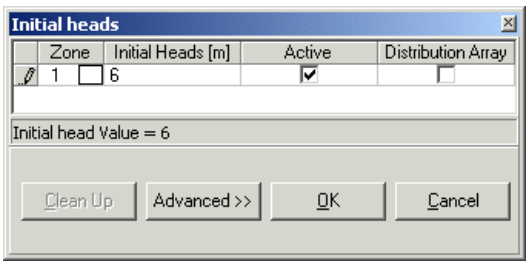


Initial Heads

The default initial heads are 6 throughout the model. To change this

- ☞ **Properties>Initial Heads** from the menu
- ☞ **[Yes]** to save the changes
- ☞ **Database** from the side menu

The following dialogue will load:



- For **Initial Heads (m)**

Type: 0

- ☞ **[OK]** to save changes and close the dialogue

Constant Head boundary

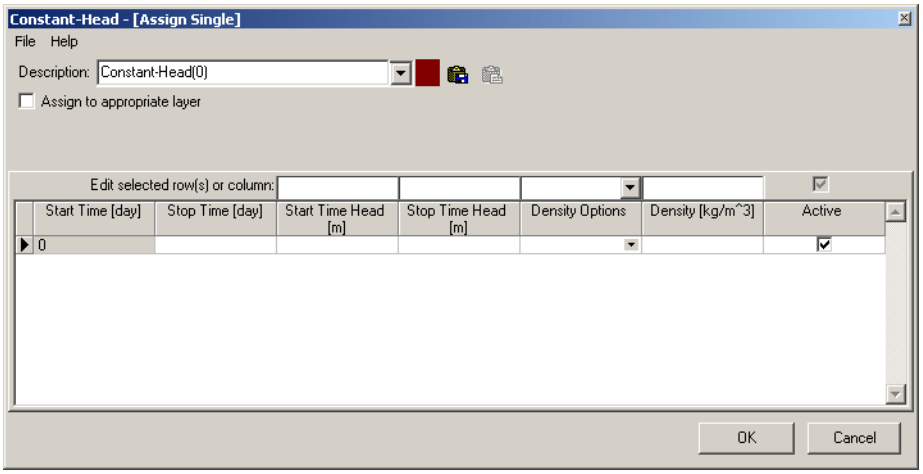
There are constant head boundaries in the Layer 2 at both sides of the model, with the value of 0 m, to provide a fluid outlet. To assign these boundaries,

- ☞ **Boundaries>Constant Head (CHD)**
- ☞ **[Yes]** to save the changes
- ☞ **Assign>Single** from the side menu

A blank Constant Head dialogue will appear with a message: “Click on cell(s) to be assigned”.

- ☞ Row1-Column1-Layer2 (move the dialogue if it is in the way)

The cell will be highlighted and the dialogue will change as follows:



☞ Row1-Column44-Layer2

This cell will also become highlighted. In the **Constant Head** dialogue,

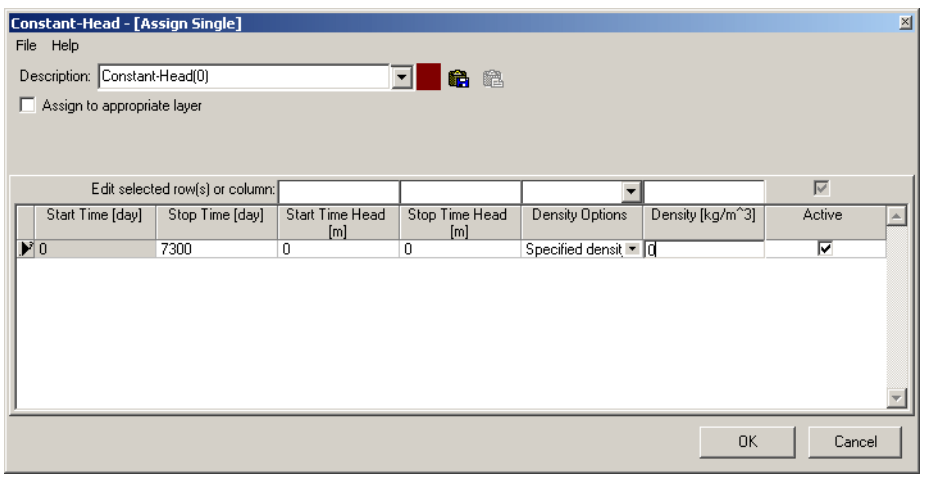
Type: 7300 in **Stop Time (day)**

Type: 0 in **Start Time Head (m)**

Type: 0 in **Stop Time Head (m)**

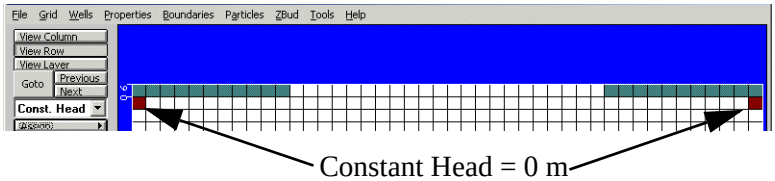
Select **Specified Density** from the **Density Options** combo box.

Type: 0 in **Density**



☞ **[OK]** to save the changes and close the dialogue

The top of your model should now look similar to the following:



The model is now set up will all of the conditions needed to run the simulation.

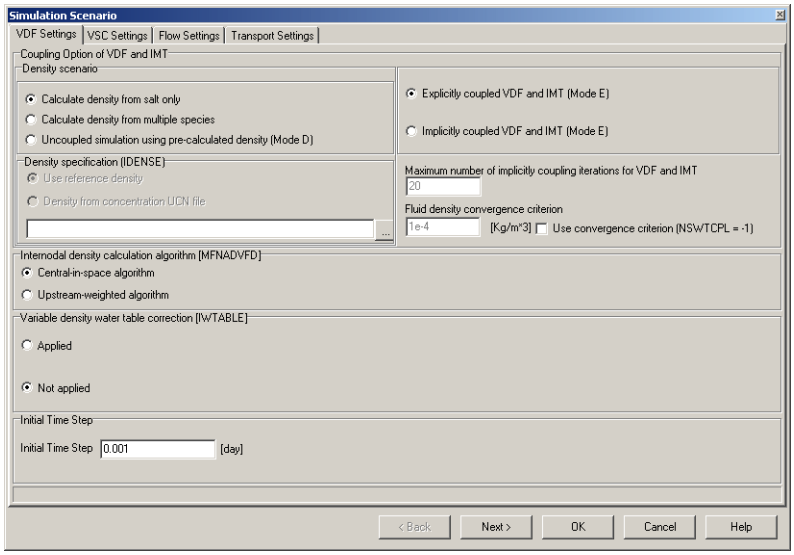
- ☞ **File>Save** to save the changes to your model
- ☞ **File>Main Menu**, or press **F10-Main Menu** button below the model domain.

SEAWAT Set-up

You will now set up the SEAWAT Simulation Scenario.

- ☞ **Run** from the top menu bar
- ☞ **SEAWAT >Simulation Scenario** from the menu

The following dialogue will load:



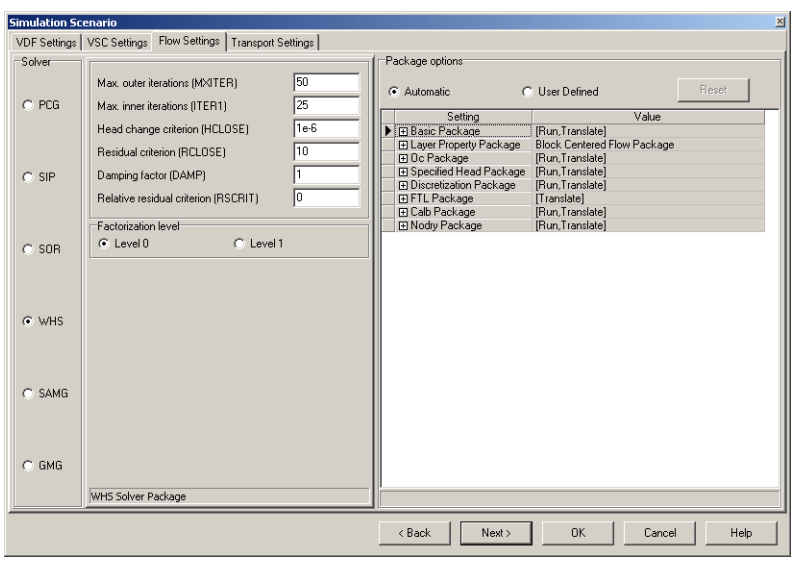
☞ **Upstream-weighted-algorithm** radio button, in the **Internodal density calculation algorithm** frame

Type: 3 in the **Initial Time Step (day)**

☞ **[Next >]**

☞ **[Next >]**

The following dialogue will load:



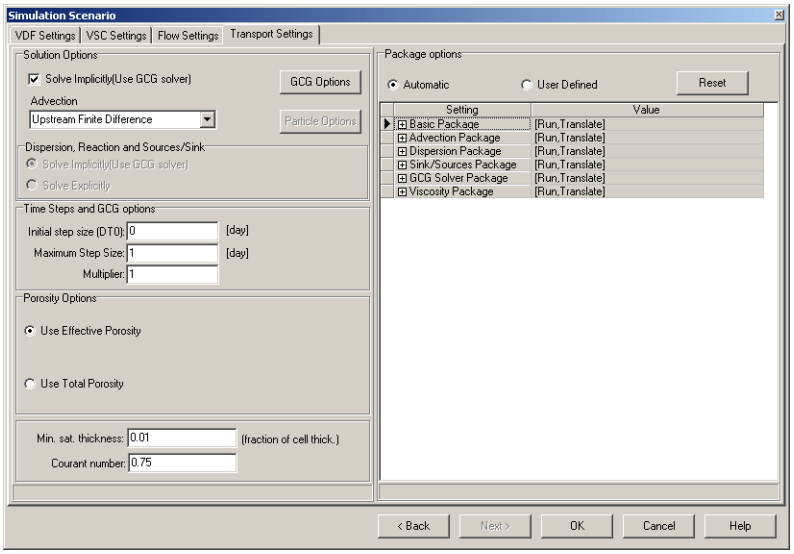
In this dialogue:

- Set the **Solver** to PCG
- Set **Max. outer iterations (MXITER)** to 20
- Set **Head Change Criterion (HCLOSE)** to 0.001
- Set **Residual Criterion (RCLOSE)** to 1
- Set **Printout interval (IPRPCG)** to 5

Leave the rest of the settings as default

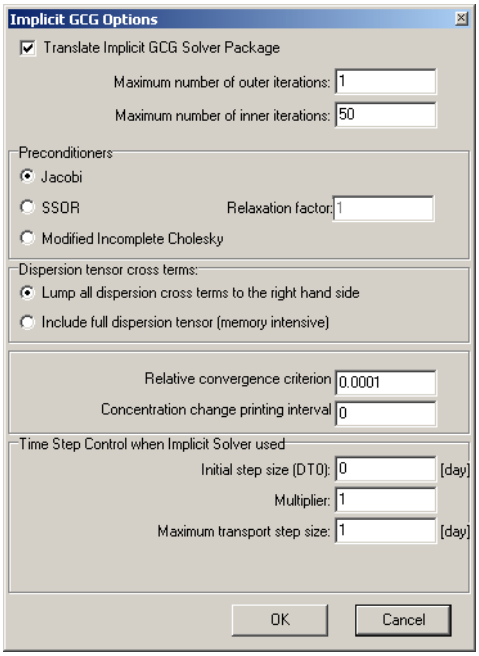
☞ **[Next >]**

The following dialogue will be loaded:



☞ **[GCG Options]** in the **Solution Options** frame

The following dialogue will load:



☞ **Modified Incomplete Cholesky** radio button in the **Preconditioners** frame

Type: 1E-8 in **Relative convergence criterion** field

Type: 5 in **Concentration change printing interval** field

Type: 3 in the **Initial step size** field

Type: 3 in the **Maximum transport step size** field

☞ **[OK]** to accept the changes and close the **GCG Options** dialogue

☞ Select **Central Finite Difference** from the **Advection** combo-box

☞ **Use Total Porosity** radio button in the **Porosity Options** frame

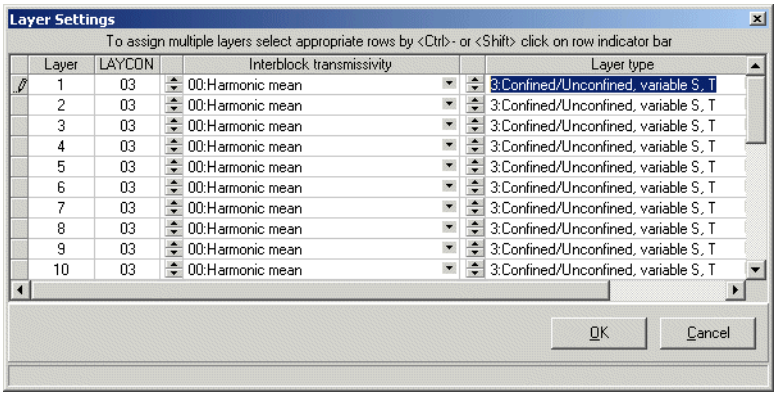
Type: 0.1 in the **Courant Number** field

☞ **[OK]**

Next, define the layer types:

☞ **SEAWAT >Flow>Layers**

The following dialogue will load:



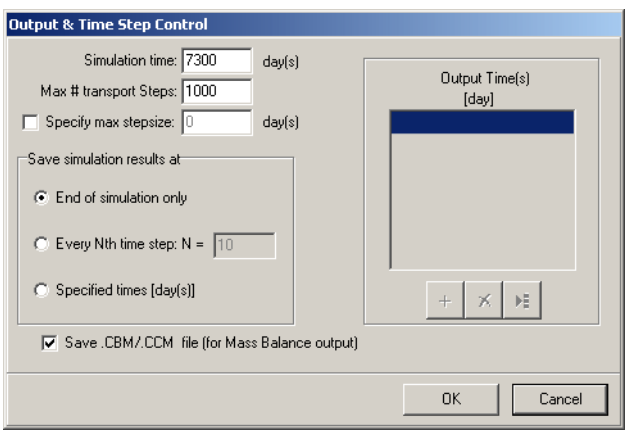
☞ On each cell in the **Layer Type** column and change the current layer type to "0:Confined, Constant S, T"

☞ **[OK]** to accept the changes and close the dialogue

Next, define the transport settings:

☞ **SEAWAT 2000>Transport>Output/Time Steps**

The following dialogue will load:



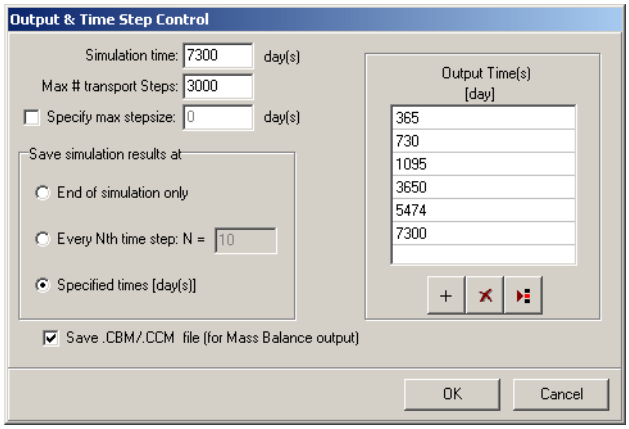
Type: 3000 in **Max. # of Transport Steps** field

☞ Specified **Times [day(s)]** from the **Save simulation results at** frame

☞ In the **Output Times** text field

Type: 365 **<Enter>**, 730 **<Enter>**, 1095 **<Enter>**, 3650 **<Enter>**, 5475 **<Enter>**, and 7300

When you are finished, the dialogue should look similar to the following:



☞ **[OK]** to accept the changes and close the dialogue

You are now ready to run the model.

Running SEAWAT

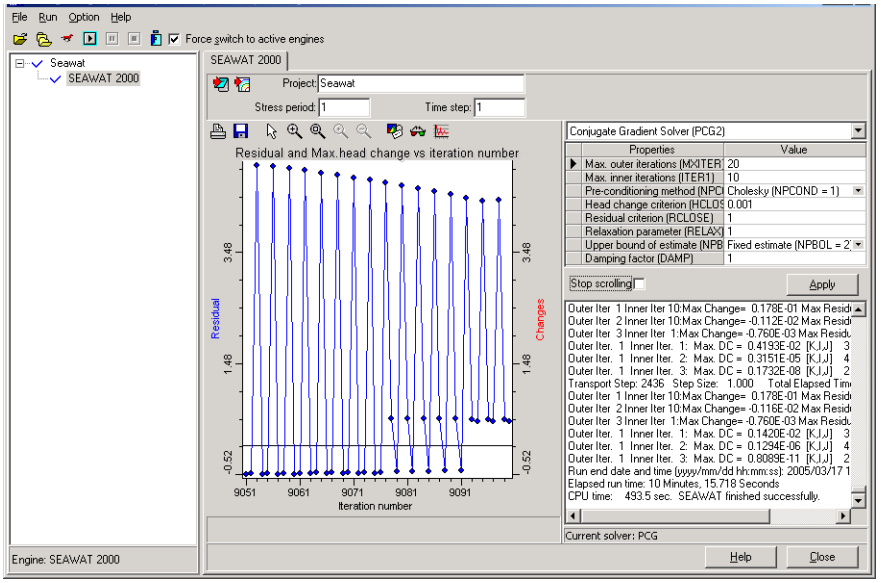
To run the model simulation,

☞ **Run** from the menu bar

☞ In the check box beside **SEAWAT**

☞ **Translate and Run**

The **VMEngine** dialogue will open which will track the progress of the simulation.



NOTE: SEAWAT requires longer run times as the complexity of the model increases. On a machine with a PentiumII processor this model requires approximately five minutes to run.

NOTE: A SEAWAT model takes about 2-4 times longer to run than a MODFLOW model of same dimensions due to the couple flow and transport.

Once the simulation has finished running,

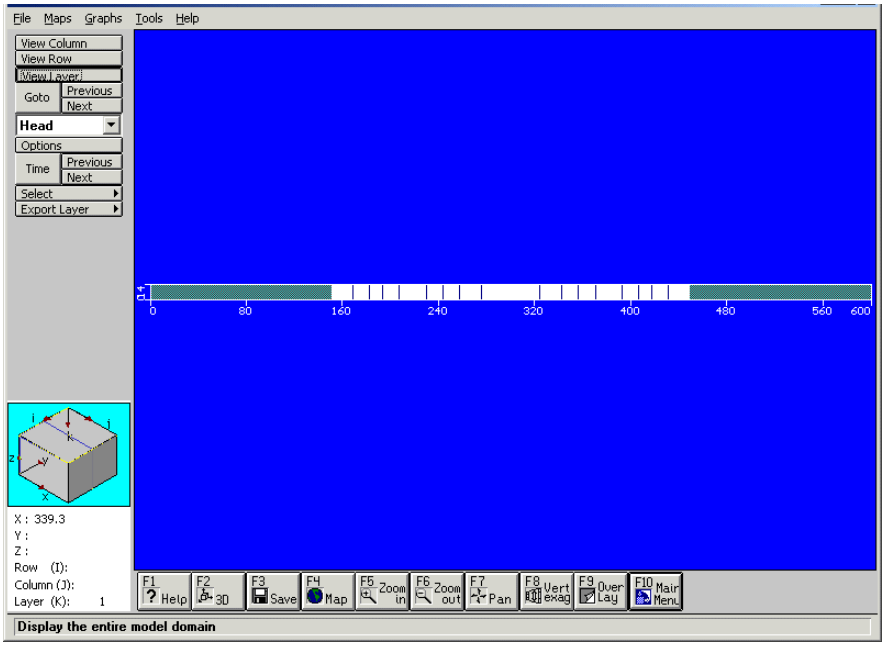
☞ **[Close]**

Viewing Output

The simulation information has been saved at the specified times as defined in **Transport & Time Step Control** dialogue described above. To view the output

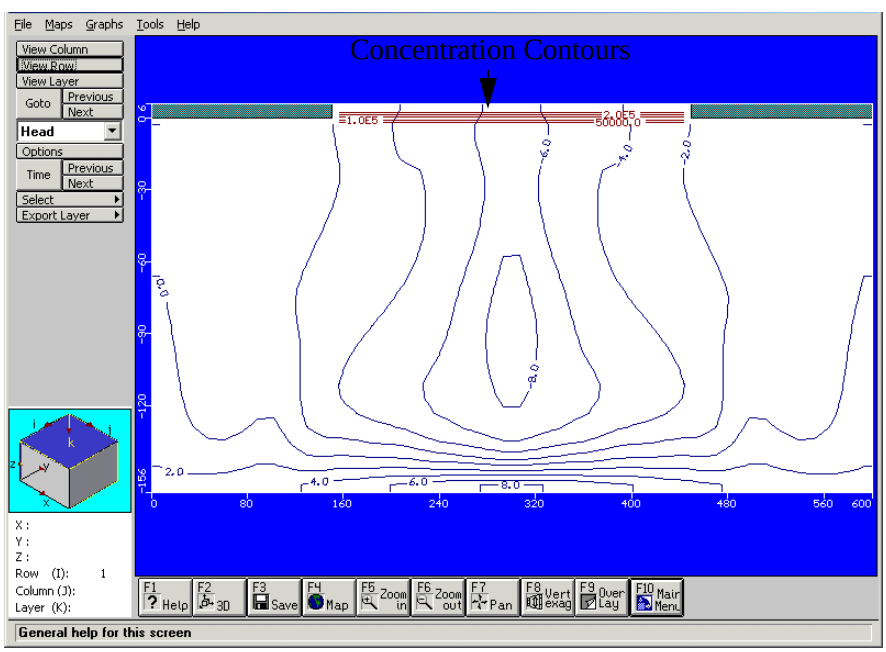
☞ **Output** from the Main Menu

The following view will be presented:

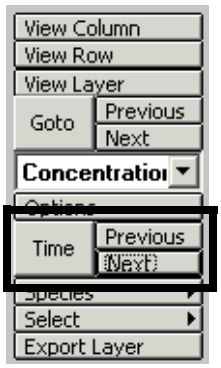


☞ **[View Row]** from the side menu

Move the mouse pointer over the model and select a row to view. You will be presented with a cross-section view of the model:



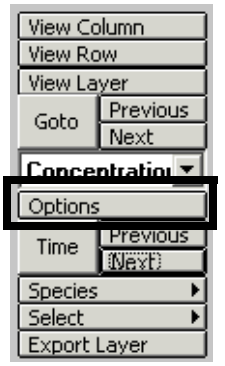
- **Maps>Contouring>Concentration** to activate the concentration contours
- **[Next]** in the side menu beside **[Time]** to advance the contours to the next time period.



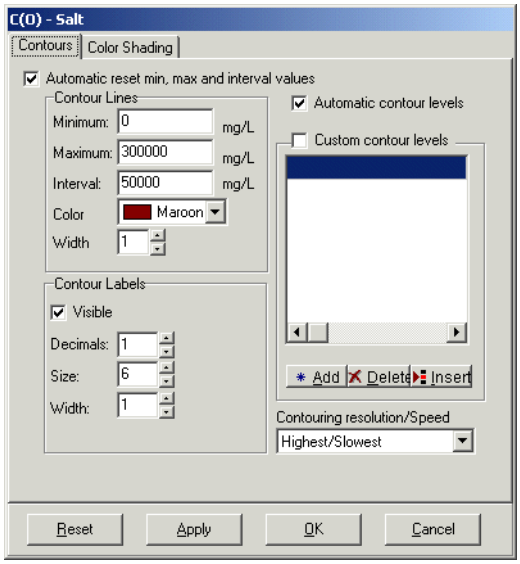
Watch as the contours trace the advancement of the concentration front.

You can also display the contours as a color-shaded gradient.

- **Time** button and select **365** to return to the first time period
- **Options** button (directly above the **Time** button)

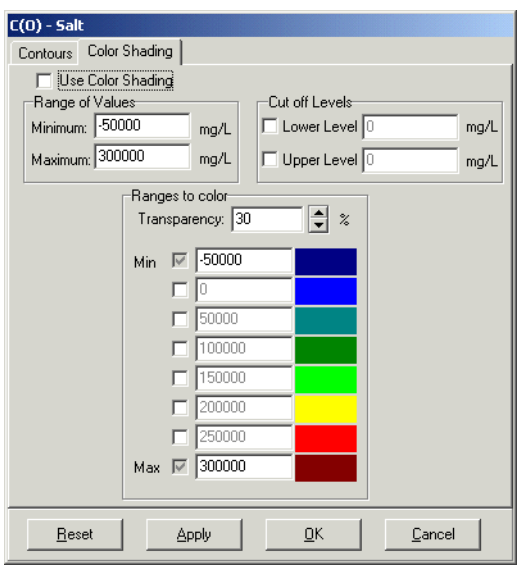


The following dialogue will load:



☞ **Color Shading** tab

The following dialogue will load:

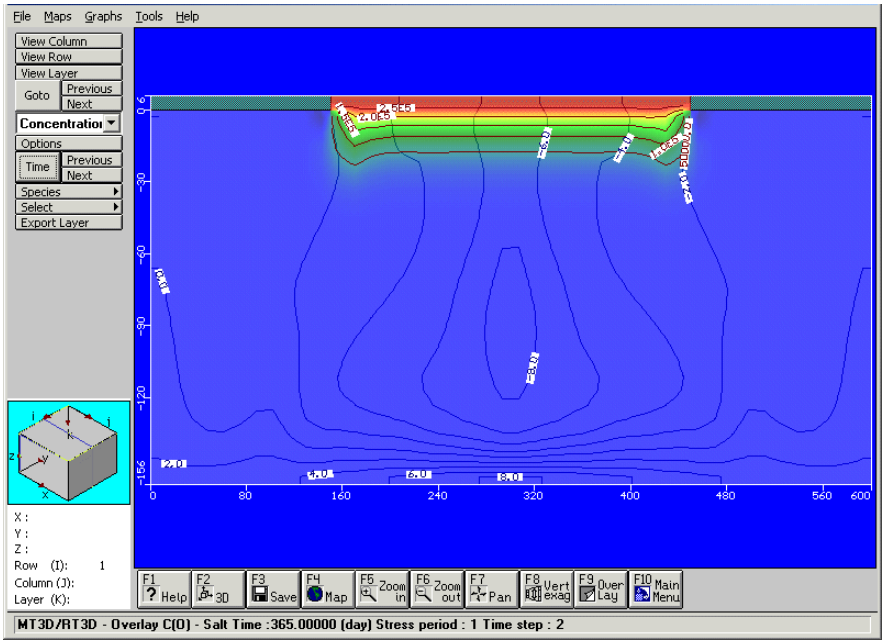


☞ **Use Color Shading** check box

Leave the rest of the settings as default

☞ **[OK]** to apply the changes and close the dialogue

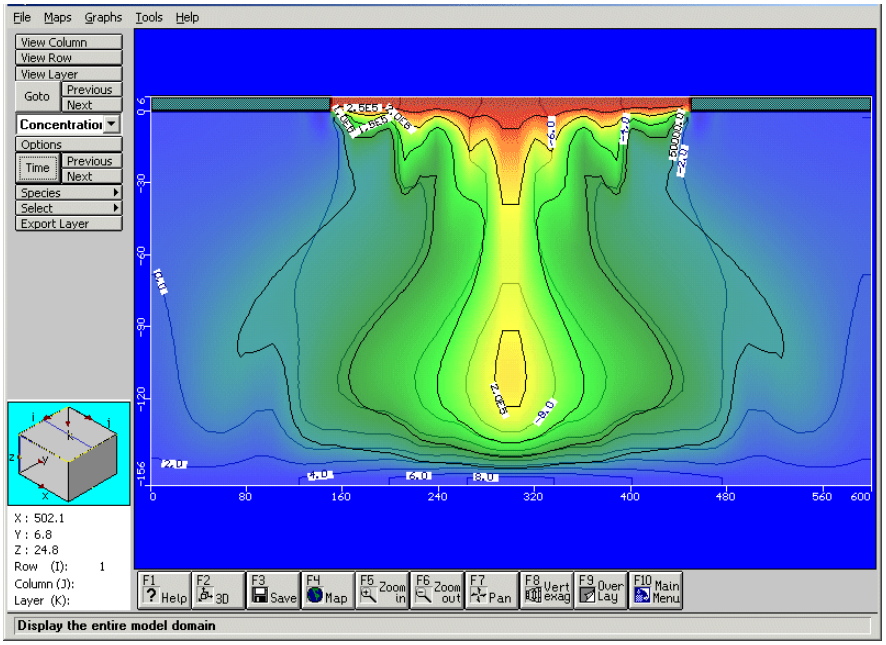
The contours should now look as follows:



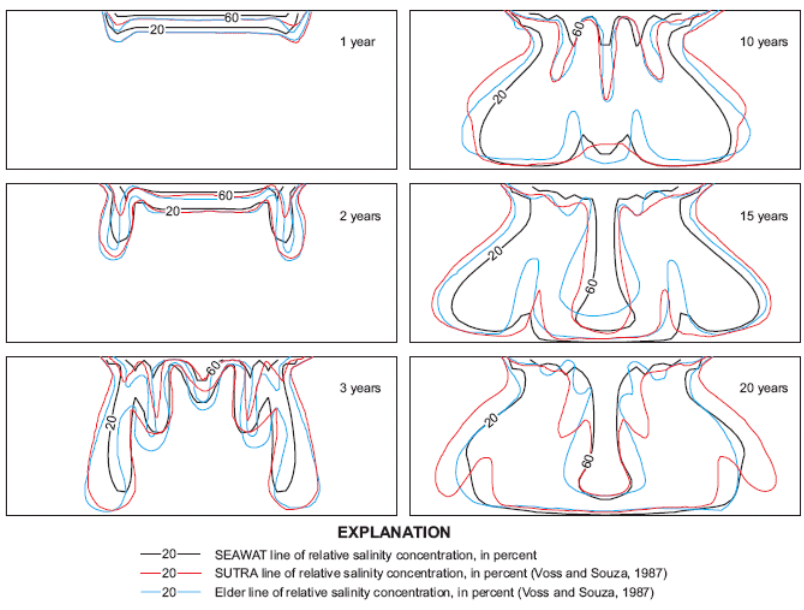
The above image shows the concentrations as they would be after 1 year (365 days). To see what the concentrations will be at the end of the 20 years (7300 days),

- ☞ **[Time]** from the side menu
- ☞ Select **7300**
- ☞ **[OK]**

The output contours will change as follows:



Compare these contours to the following published results:



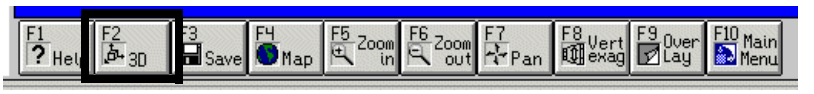
Comparison between SEAWAT, SUTRA, and Elder's solution for the Elder problem over time.

Note that in the published results the concentrations are displayed as a percentage of the constant concentration at the top of the model. In the default Visual MODFLOW output the concentrations are displayed as absolute values.

You can print any current view by clicking the **File>Print** from the menu. This will launch the **Image Editor** window which allows you to make certain modifications to your print-out such as adding a title and/or a legend. When you are satisfied with the view, click button from the tool bar.

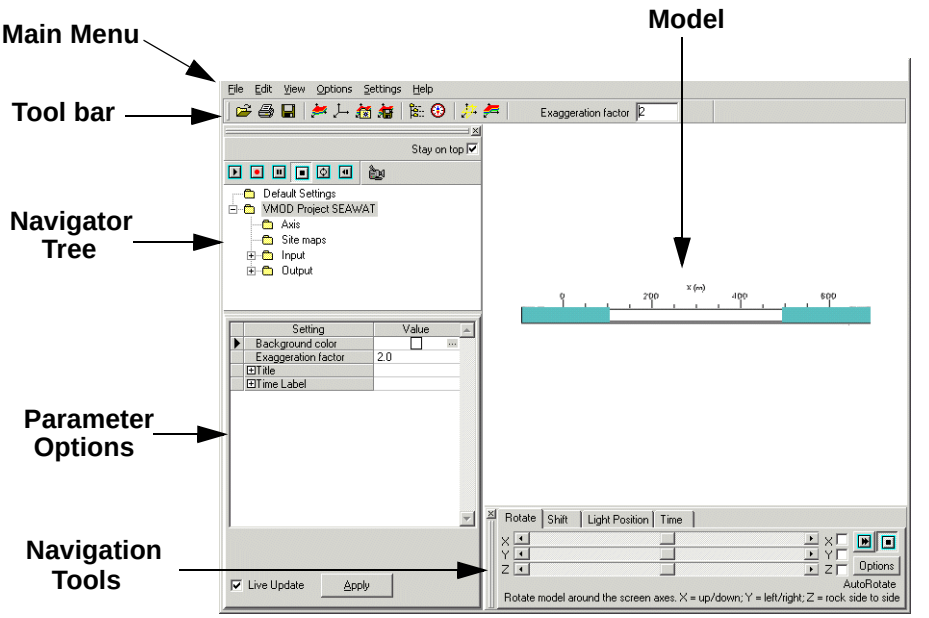
You can also see this advancement in **3D Explorer**. To do so

☞ **F2 3D** button below the model or press F2 button on your keyboard



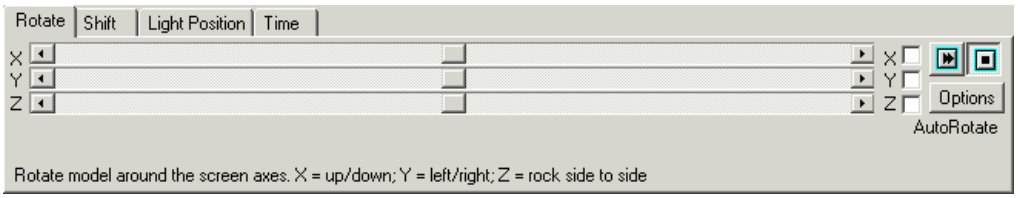
Visualizing the model with 3D Explorer

The **3D Explorer** will load, allowing you to manipulate the model view:



The **Navigation Tools** at the bottom of the window allow you to rotate and shift the model, shift light position, and control time point for the model.

The default view is from the top. Use the **Rotation tools** to gain a better view of the model:



☞ Drag the **X**, **Y**, and **Z** sliders left and right to rotate the model

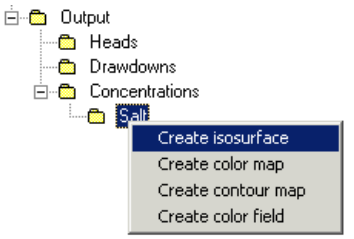
To visualize the concentrations, in the Project Navigator

☞ **[+]** beside Output

☞ **[+]** beside Concentrations

☞ **Salt**

☞ Right-click on **Salt** and choose **Create Isosurface** from the menu

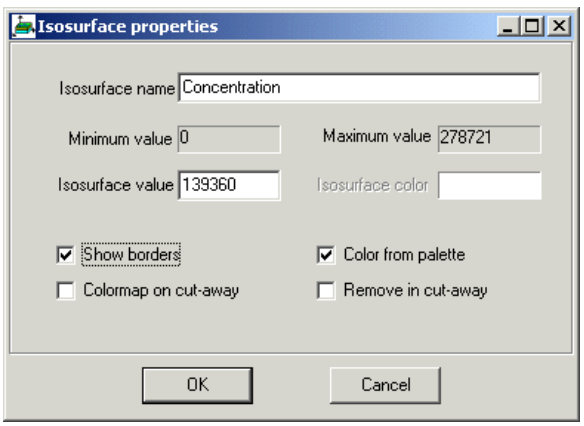


Type: "Concentration" in **Isosurface name**

Type: 10 in **Isosurface value**

☞ Select **Show borders** and **Color from palette**

Leave the rest of the settings as default:



- ☞ **[OK]**
- ☞ **Time** tab of the **Navigation tools**

Use the slider to advance the time, or

- ☞ button to view the automatic sequence
- ☞ button from the tool bar to print the model, or select **File>Print** from the main menu

Set up the image to be printed in the **Image Editor**. When you are done,

- ☞ **[X]** to close the **Image Editor** (printing dialogue)
- ☞ **[X]** to close the **3D Explorer**

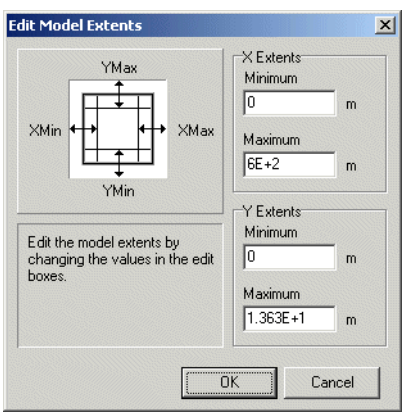
Expanding the Model

Up until now you have been working with a two-dimensional model. You can increase the number of rows and make your model three-dimensional, however the processing time will increase accordingly.

If you wish to expand your model, do so using the following procedure (this will increase the model to two rows):

- ☞ Return to the **Input** window
- ☞ **View Layer** from the side menu
- ☞ Layer1
- ☞ **Grid** from the menu
- ☞ **Edit Extents** from the side menu

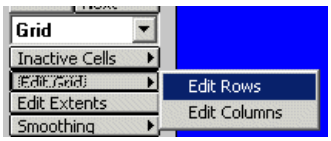
The following dialogue will load:



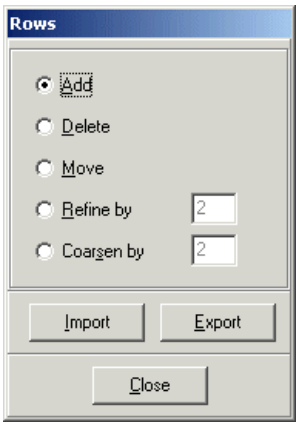
Type: 27.26 in **Y-Extents** frame, **Maximum** field

☞ **[OK]** to save the changes and exit the dialogue

☞ **Edit Grid>Edit Rows** from the side menu



The following dialogue will load:



☞ **Refine by** radio button

Since you wish to refine by 2 (i.e. make twice as many rows in the model as you have now) you can leave the default value in the field, however, if you wish to refine by some other amount, type in the desired value in the adjacent field.

Move the cursor over the model until the top LINE is highlighted in pink.

☞ On the highlighted line

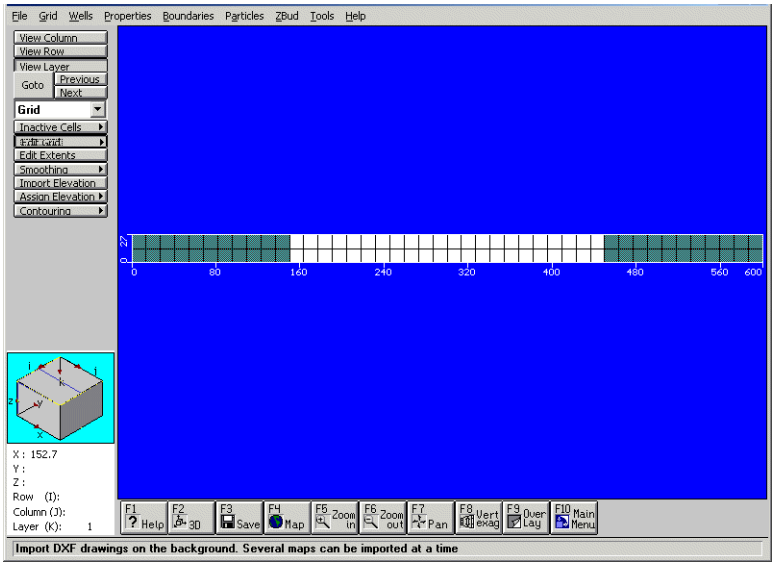
Move the cursor over the model again until the bottom LINE is highlighted in pink

☞ On the highlighted line

This procedure is used to denote the extents of the region to which you wish to apply the changes. Once this is done, a new line will be drawn, separating the one row into two.

☞ **[Close]** to close the dialogue

Your model should now look as follows:



You can run this expanded model by returning to the **Run** window and selecting **Run>Translate&Run** from the menu.

This concludes the SEAWAT tutorial.

References

Langevin, C.D., Shoemaker, W.B., and Guo, W. 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 Processes (IMT). U.S. Geological Survey. Tallahassee, Florida