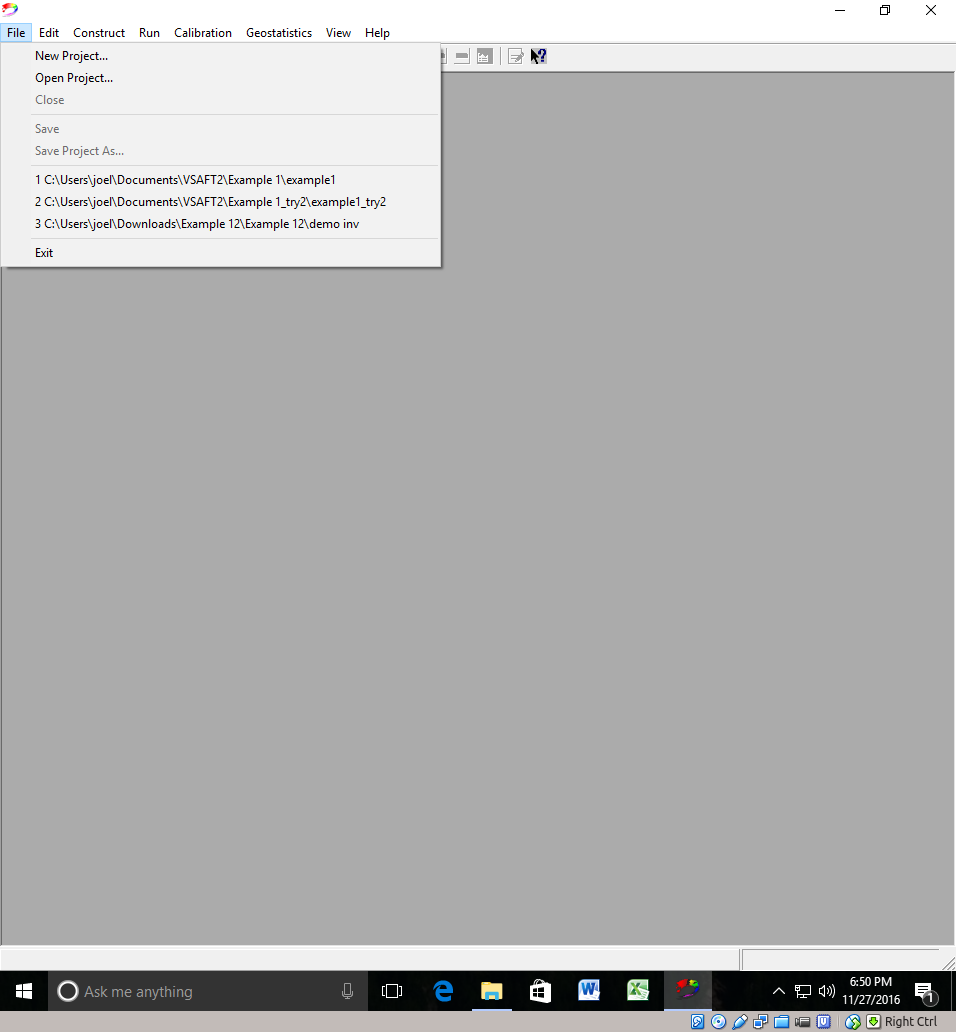
**Example 2: Vertical Transient Flow in a Homogeneous Media**

The instructions for this example are provided as a list of steps with accompanying screen captures.

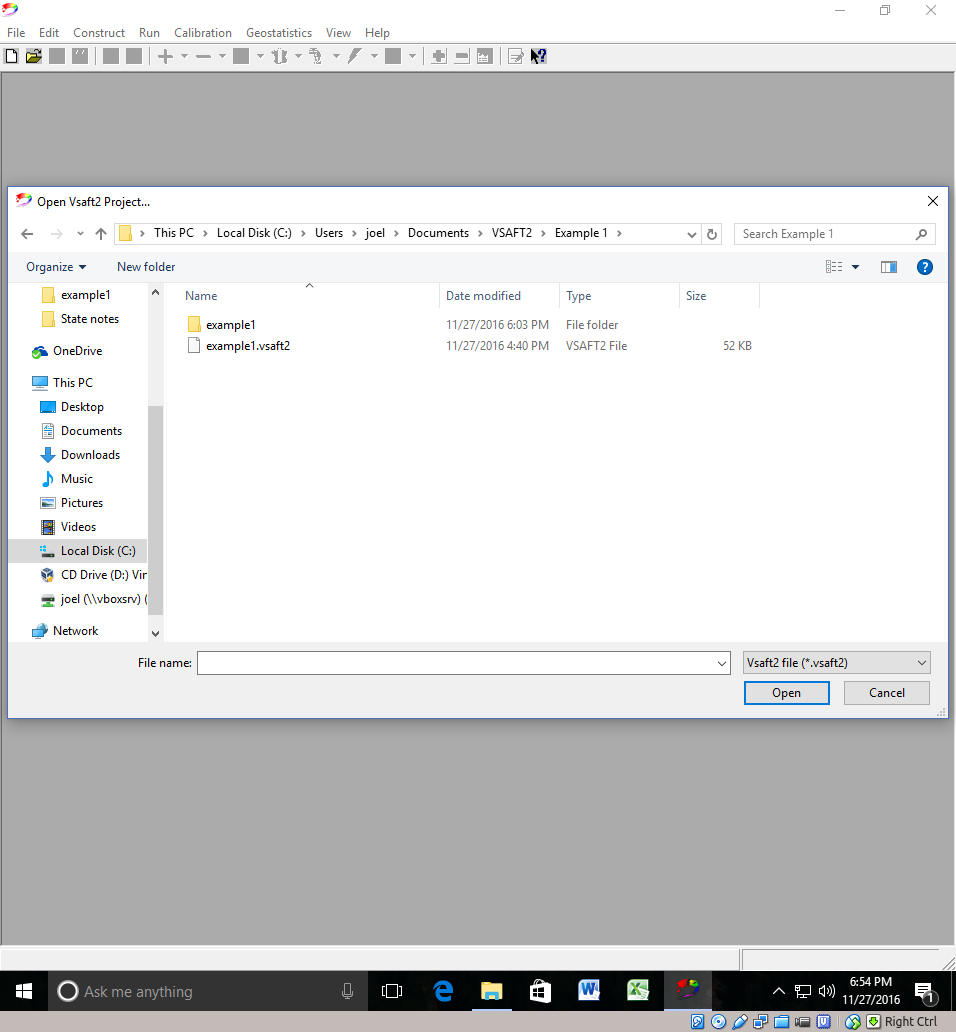
1. Open Project

* To open an existing project, select **File** then **Open Project.**

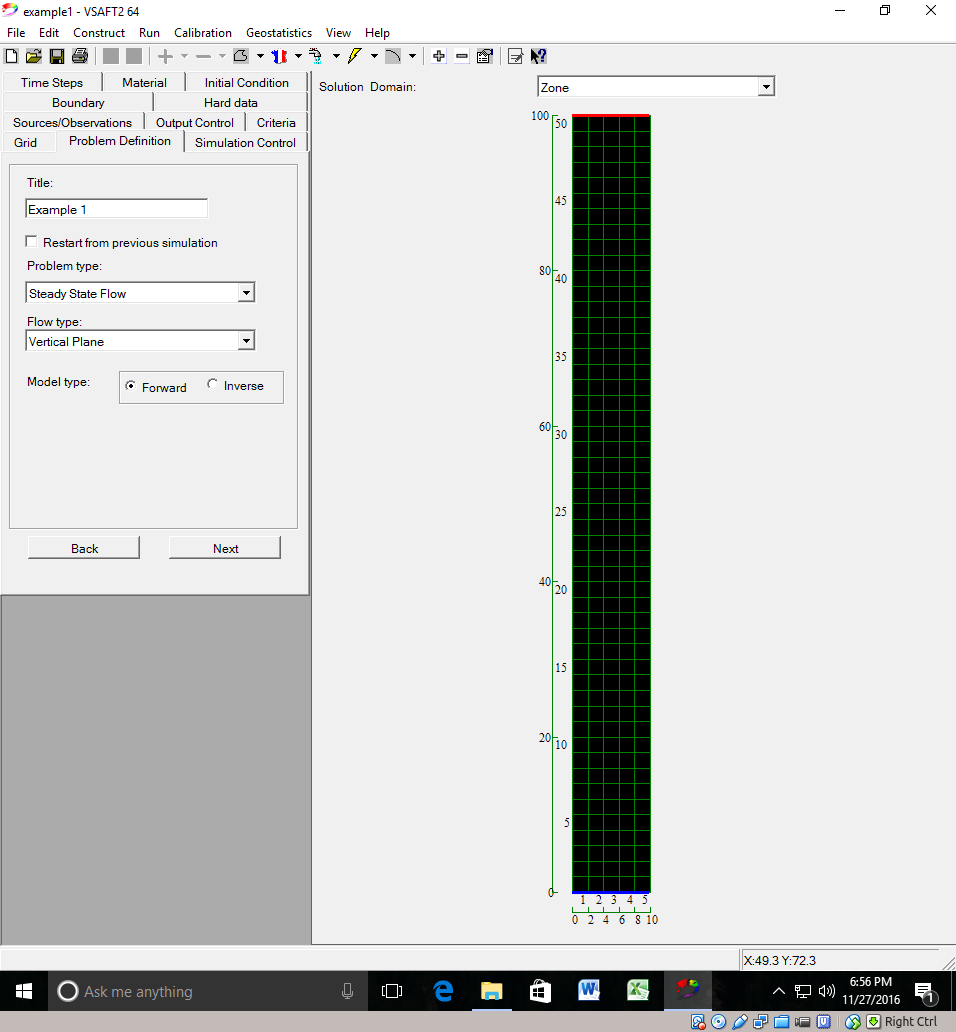


Find the example1 project directory from using the pop-up browser.

* Be sure to double click on the directory so that the project directory appears in the location window.
* Select the **example1.vsaft2** file then select **Open.**
* Once example1 project is loaded click on the **problem definition tab**.



1. Problem Definition

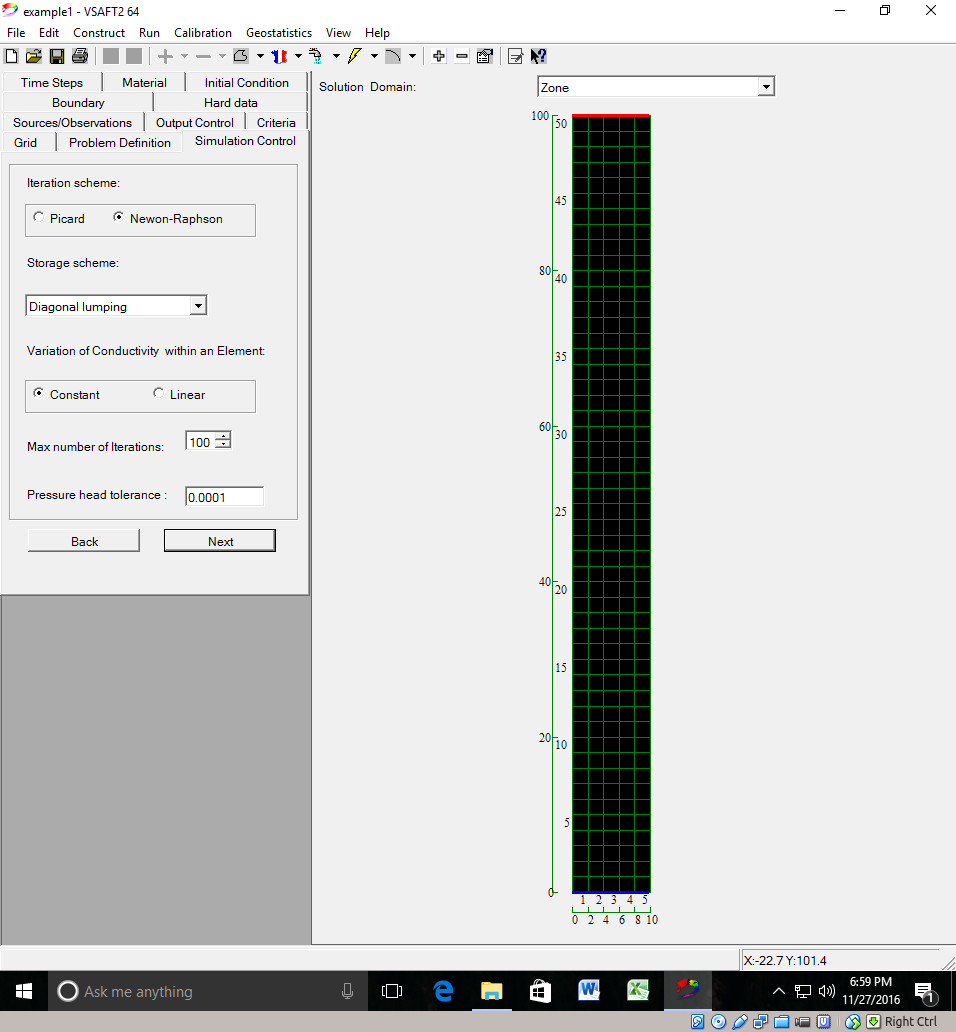


* Enter a title (i.e. Example2) in the “TITLE” box. This is for record keeping purposes and to assist in remembering the details of the model. Use a descriptive title.
* Use the “Problem type” drop down menu to select **Transient Flow.** The model has now been switched to a transient model. Keep all other parameters as the same.
* Select **Next** to continue to the “simulation control” tab .

1. Simulation Control

We will accept the default simulation control parameters for this example

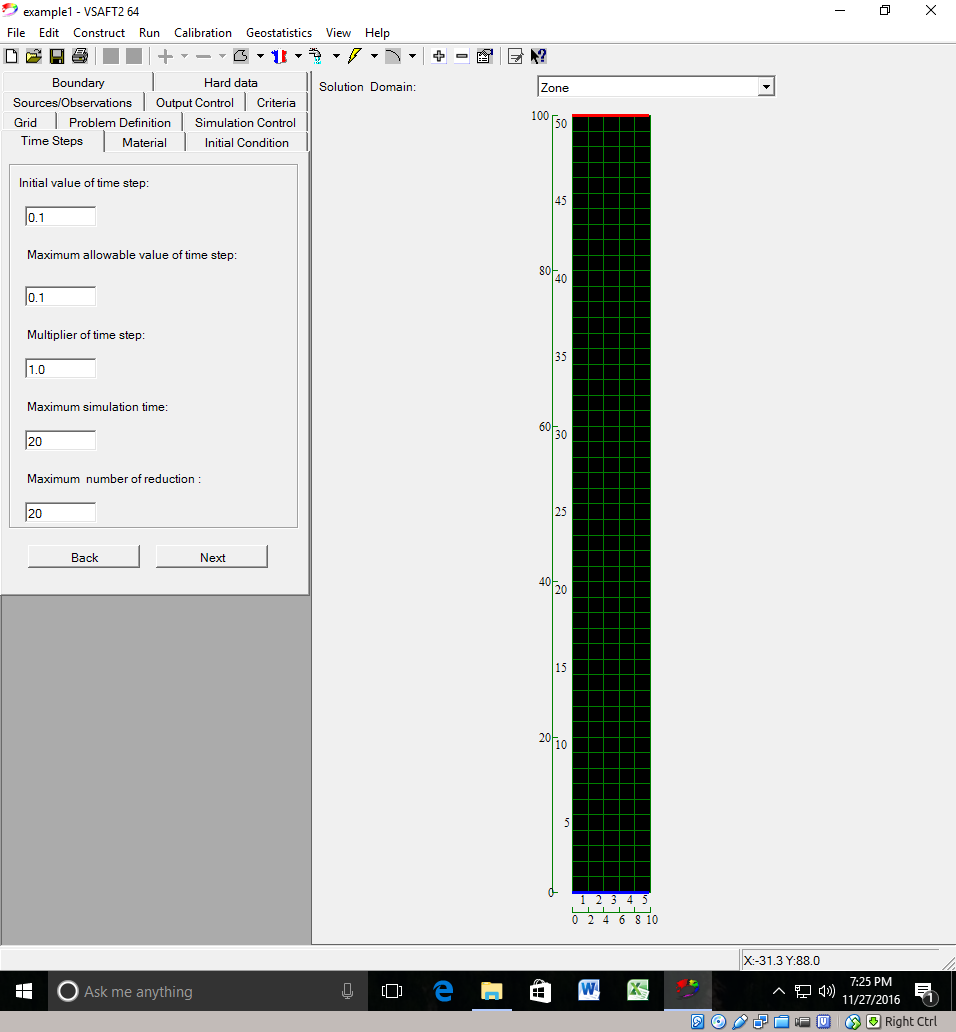
* Select **Next** to continue to the “time steps” tab.



1. Time Steps

Enter the time step parameters:

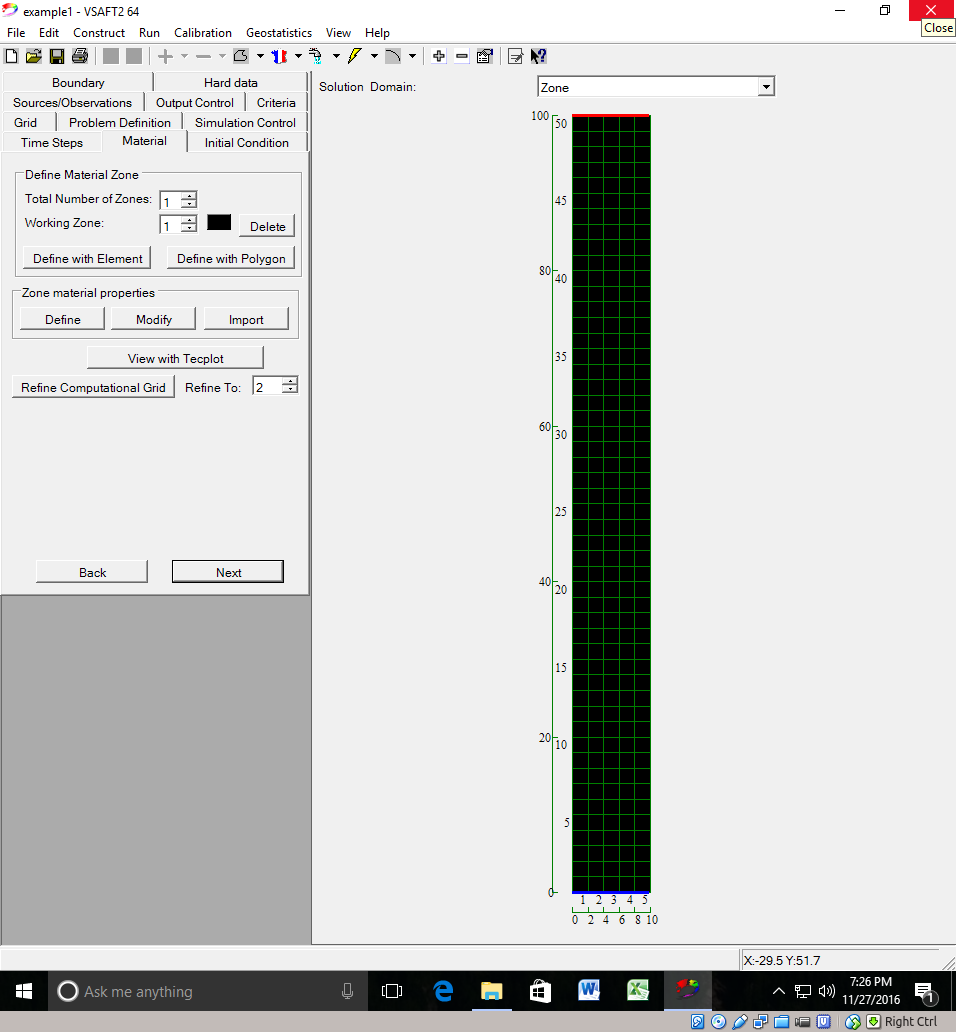
* Set the **“**initial value” of time step to **0.1.**
* Set the “maximum allowable value of time step” to **0.1.**
* Set the “multiplier of time step” to **1.**
* Set the “maximum simulation time” to **20.**
* Set the “maximum number of reductions”to **20**
* Select **NEXT** to continue to the “materials” tab.



1. Materials

The material distribution is the same as for example 1 so it does not need editing.

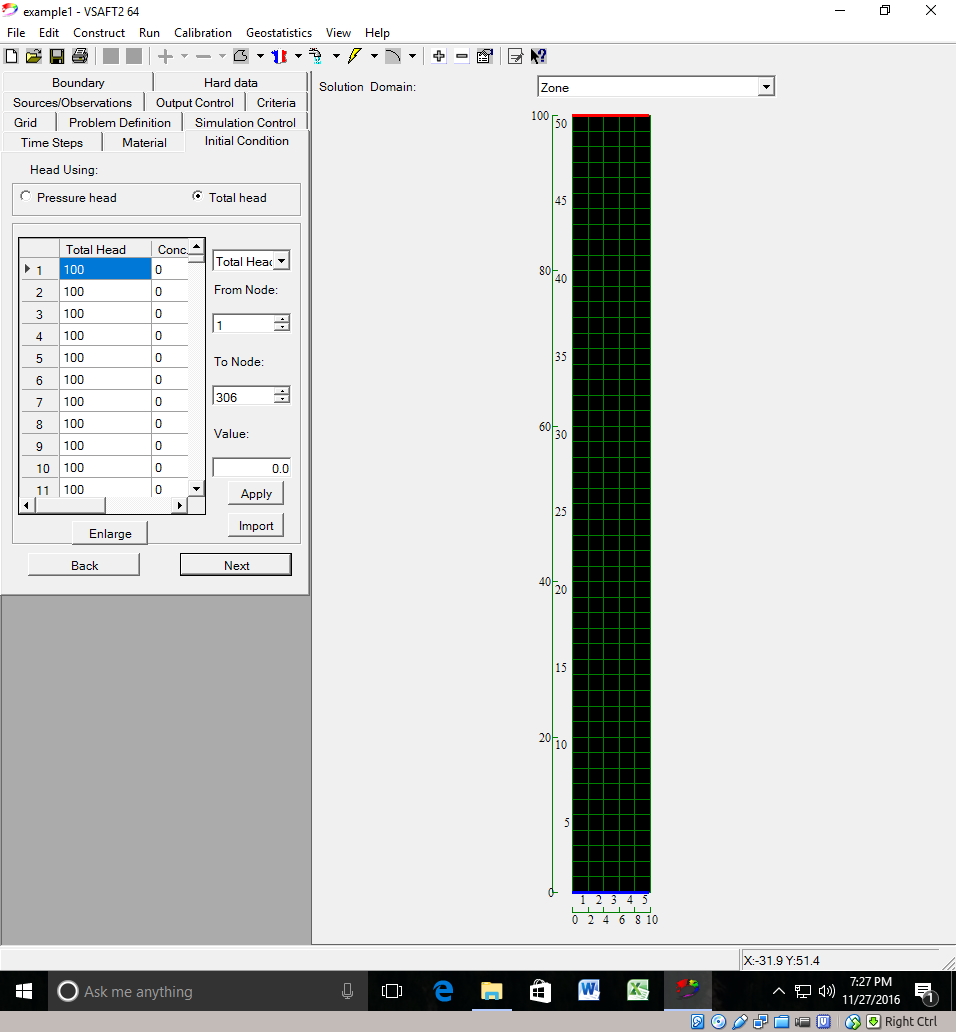
* Select **Next** to continue to the “initial condition” tab.



1. Initial conditions

The initial conditions are the same as for example 1 so they do not need editing.

* Select **Next** to continue to the “Boundary” tab.



1. Boundary

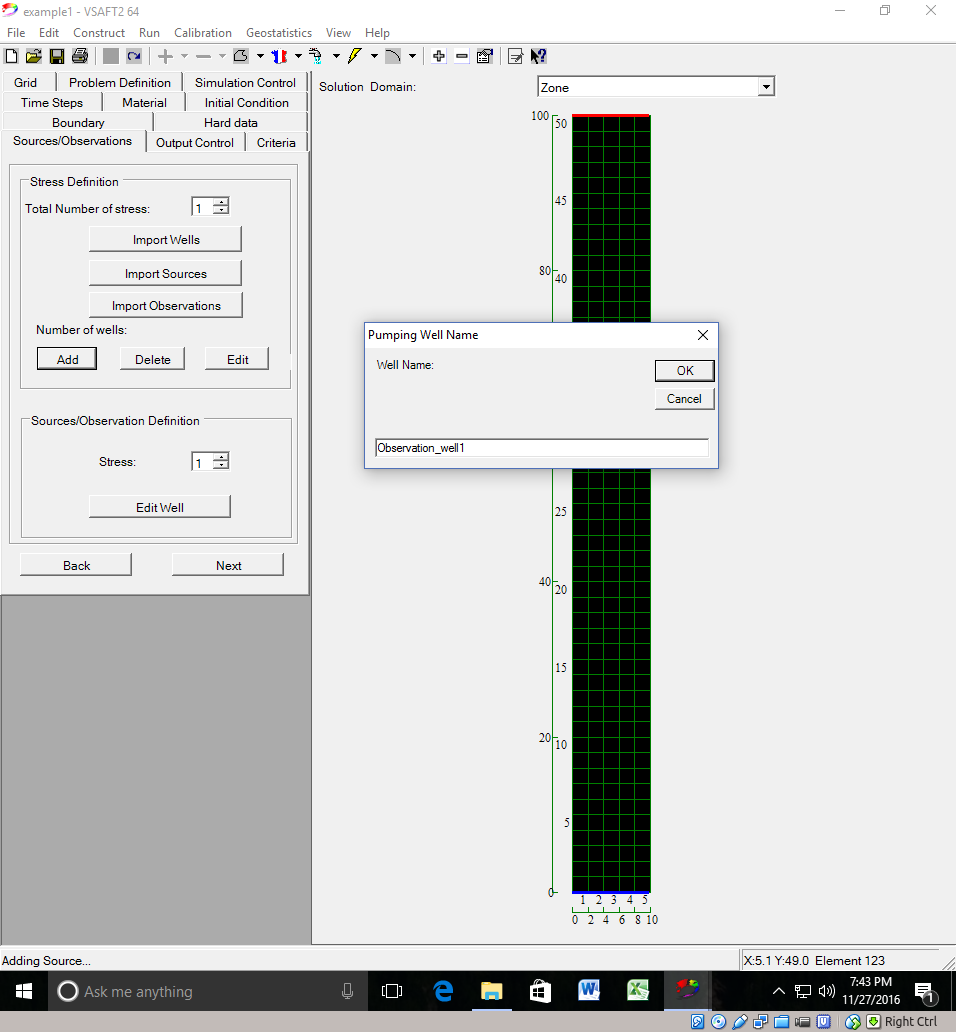
The boundary conditions are the same as example 1 so it does not need editing.

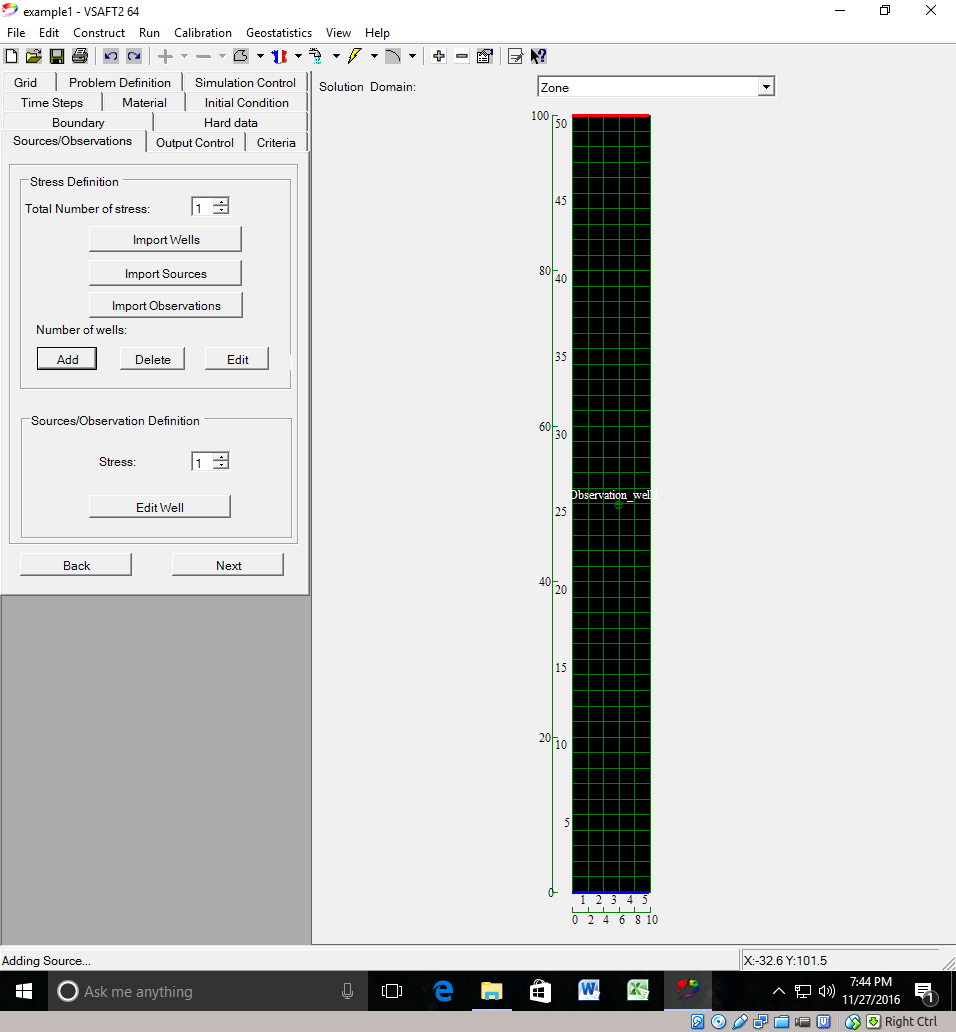
* Select **Next** to continue to the “Source” tab.

1. Sources

The sources are the same as example 1 so they do not need editing. However here we will add an observation point.

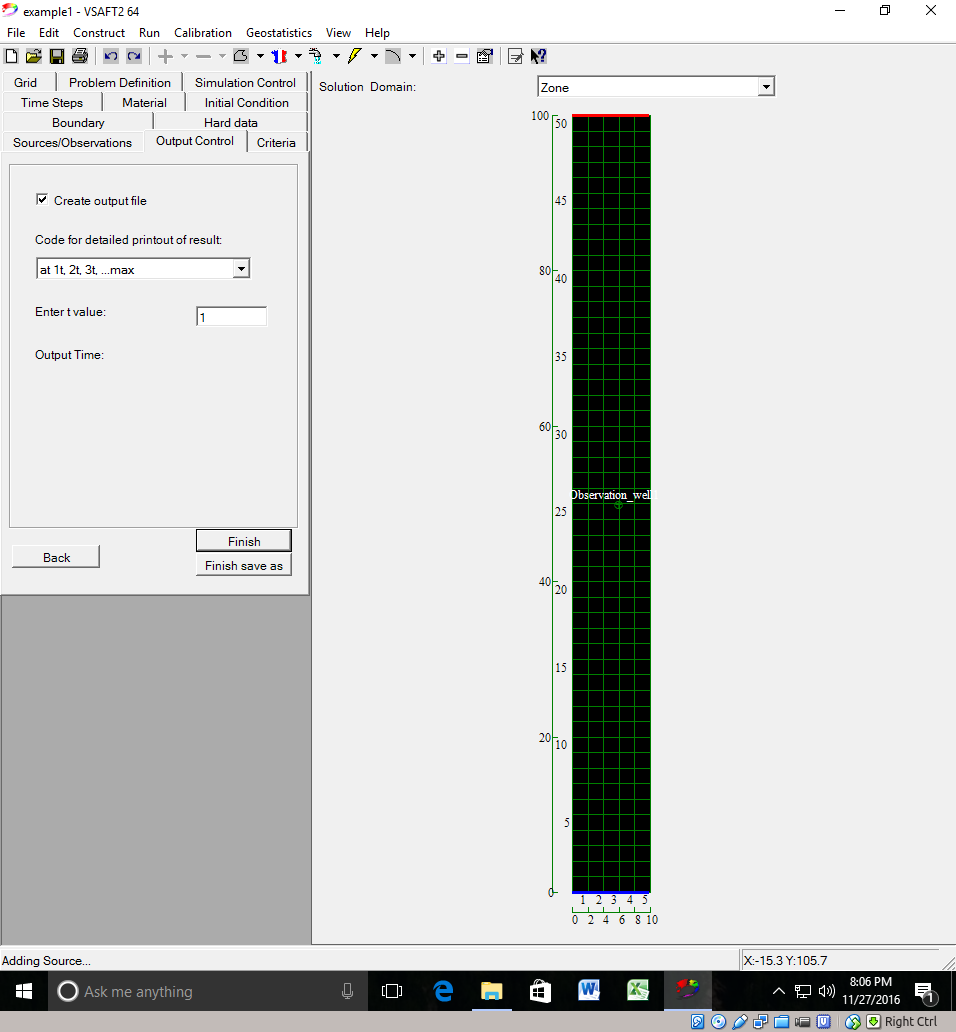
* In the “Stress Definition” section select **ADD**.
* Then move the cursor over the model domain and select the center node along **x=5 cm** and **y=50 cm**.
* Name the well as **Observation\_well1** then select **OK**. A hollow circle will appear at these points.
* Now select **Next** to move on to “Output Control”.



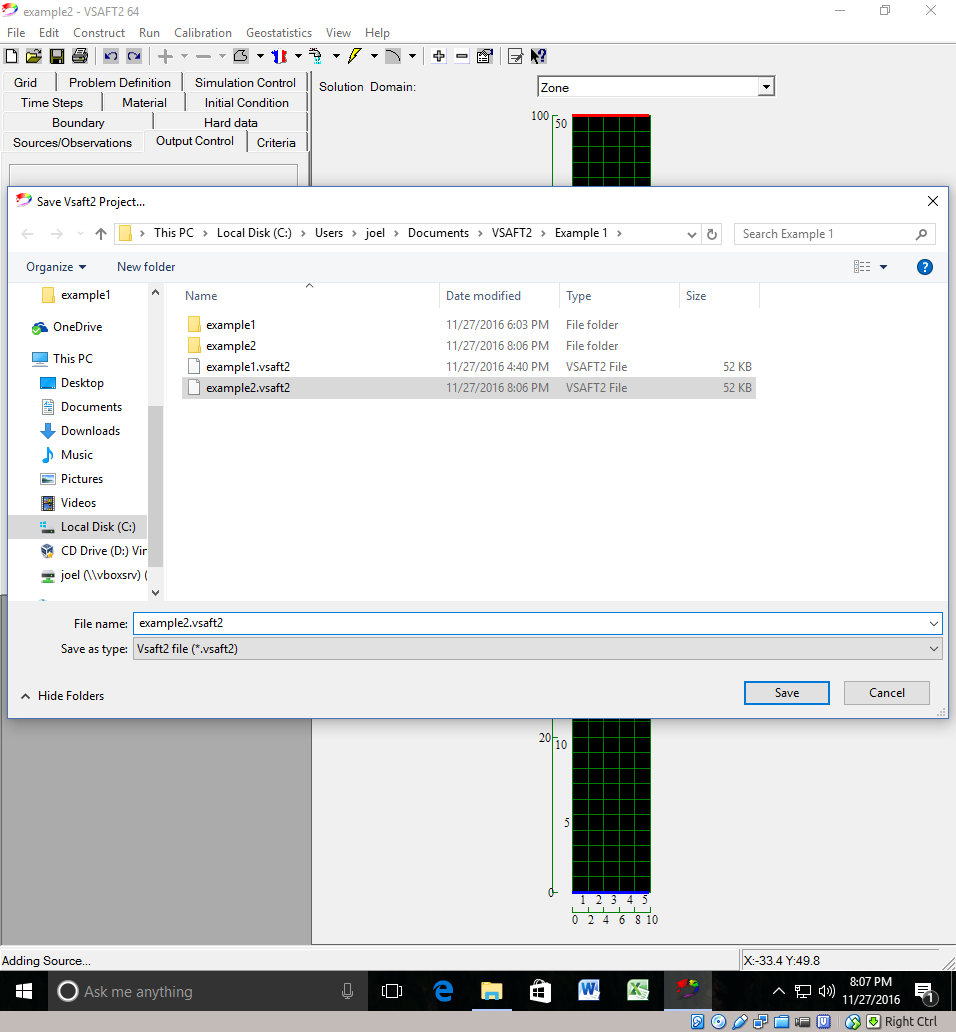


1. Output Control

* Select the time steps for the solution to be printed. Make sure there is a checkmark by the **Create Output file**.
* From the drop down menu select **at 1t, 2t, 3t,…max** and enter **1** in the in the box beside “enter t value”.



* Select **Finish save as** to finish the model setup and save the project to a directory.
* Save your project with the name **example2** and select **OK.** The model is now defined and the input files for VSAFT have been generated.



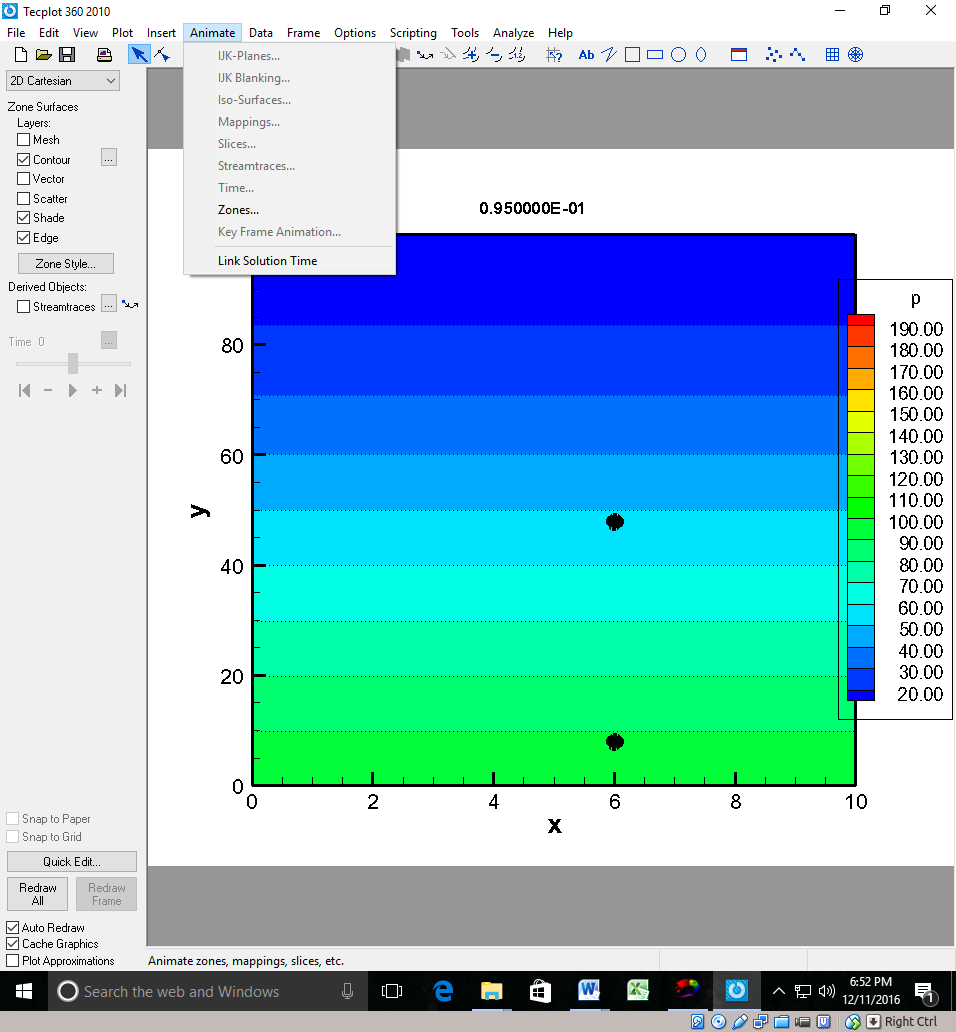
1. Running VASFT2

* Run VSAFT2 as shown in example1.

1. Viewing results in TECPLOT

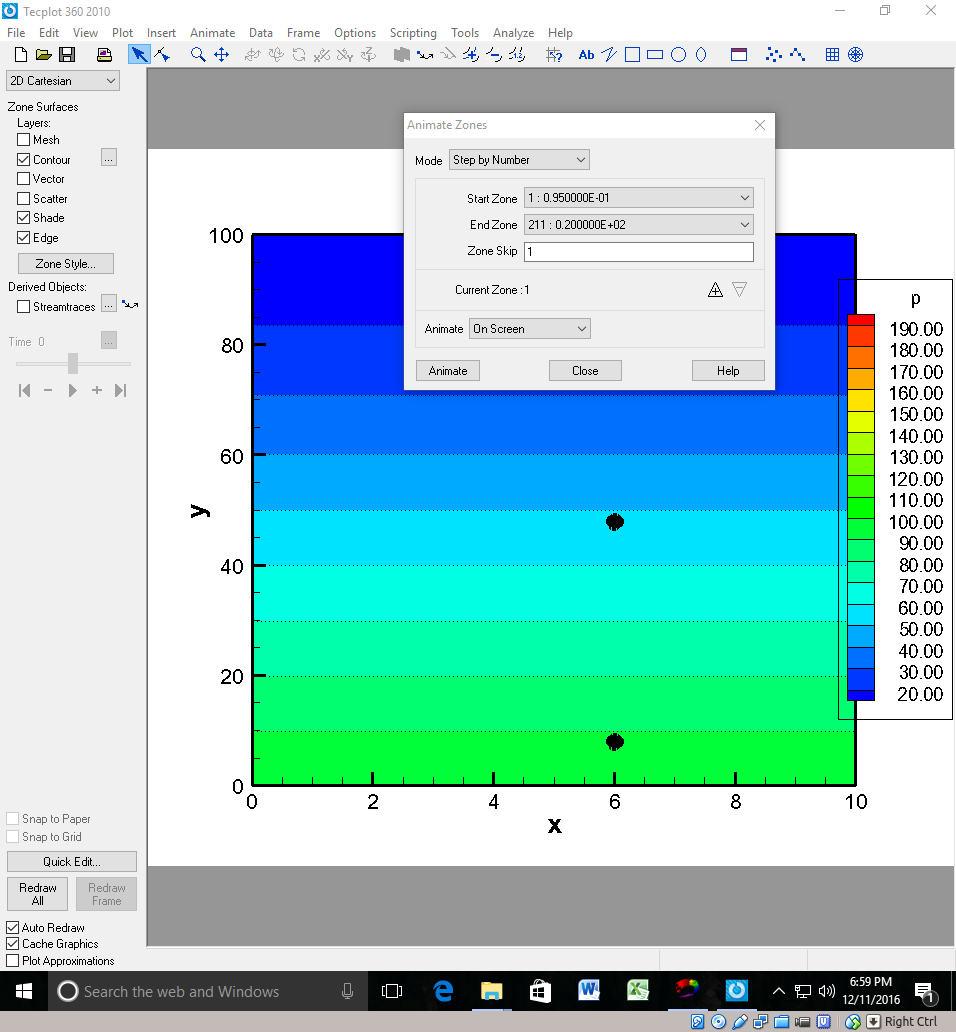
To view results, select the “View” menu in the menu bar. To view the hydraulic head distribution, select **View-> Forward->Contour->State Variable Fields**. A list dialog box for selection of the appropriate .dat file will be displayed.

* Select **Plot-Stress1.dat** which contains the hydraulic head data. This will automatically start TECPLOT and display the hydraulic head as a multi colored contour diagram.

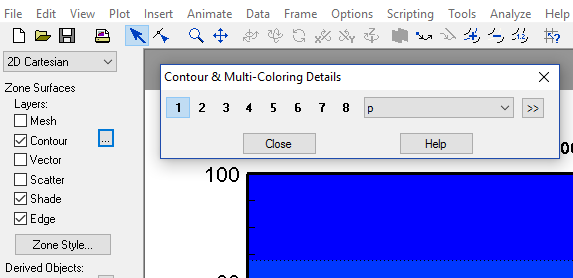


The two dots represent the observation wells added to the domain.

* We can animate the change in the equipotential lines over the time steps in the “Plot-Stress1.dat” file by selecting the “Animate” drop down menu then selecting **Zones…**
* Once the “Animate Zones” dialog box is shown, you can use it to animate the data by selecting **Animate**.



Note: The variable being using to generate the contour map can be adjusted by selecting the button to the right of the checked Contour box.

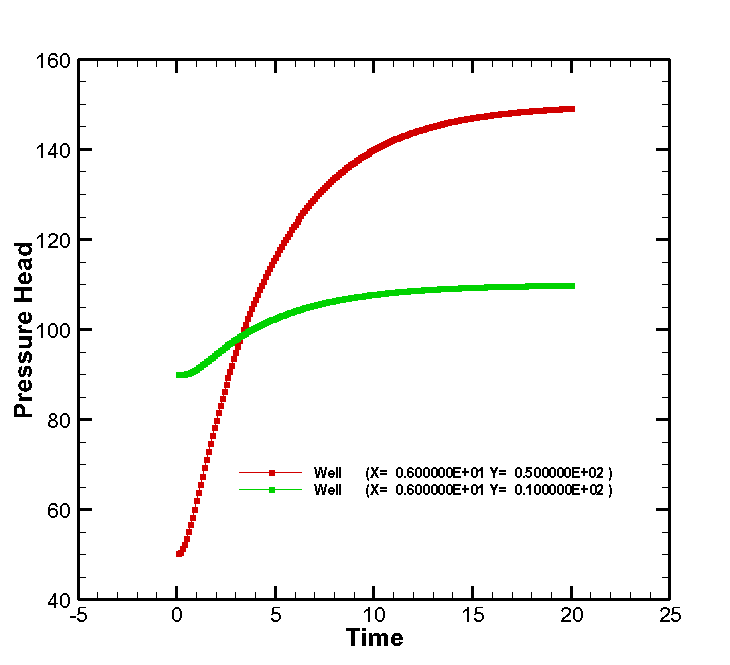


1. Viewing hydrograph

You can view well observational data such as hydraulic head change with time for the two observation wells.

* Select **View->Forward->State Variable Process** from the view menu of VSAFT2.
* A file dialogue box will open.
* Select **FwdObs-Stress1.dat** generated from running the model. This file contains all of the observable data calculated at each well site.

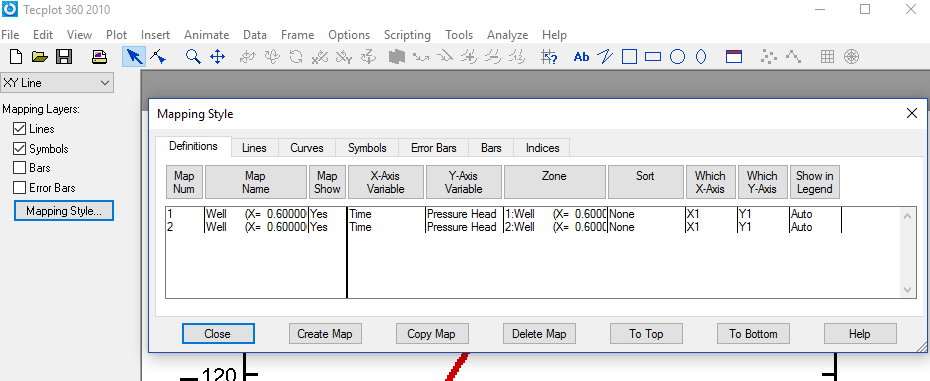
By default, pressure head vs time will be shown as seen below for both wells.



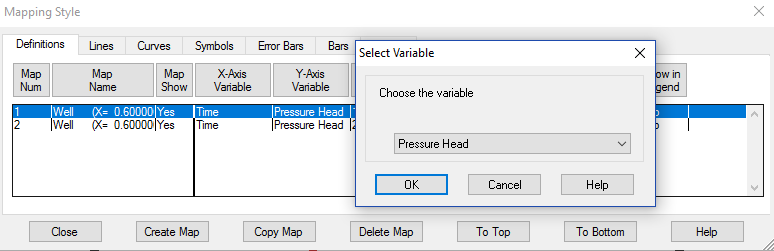
1. Generating other graphs from the observation wells

Using FwdObs-Stress1.dat we can generate graphs of other properties using Tecplot.

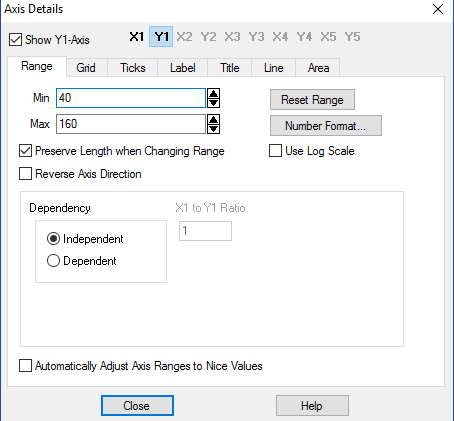
* Once you loaded the FwdObs-Stress1.dat, as described above. Select **mapping style** in the Tecplot window.



* The “mapping style” dialog box is shown below. Select one of the wells, in this case **line-1**, shown in the “Map Num” section.
* Once 1 is highlighted click **Y-Axis Variable**.

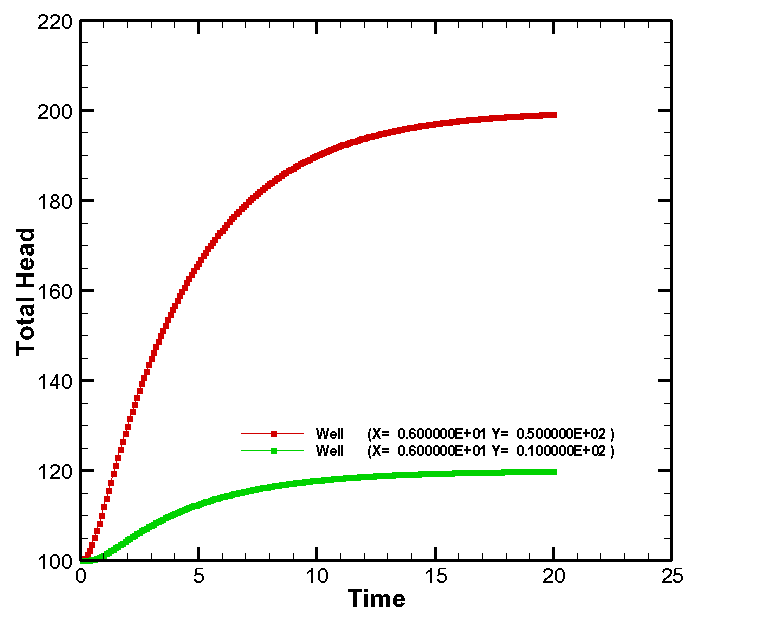


* The “Select Variable” dialog box will be displayed. You can choose to plot a variety of variables measured at each well. Choose **Total Head.**
* With a change of variables you will need to adjust both axes to best display the data. Click **OK** then select **Close** in the mapping style dialog box.
* In the main window select **Plot->Axis…**



* The “Axis Details” dialog box will be displayed. Click on **Y1** to edit the y-axis.
* Click on **Reset** **Range** to adjust the axis to fit all of the data. You can repeat this process for the x-axis.

Your final graph should look something like the one below for Total Head.



As another example, using a similar method as described above, you can generate a flux vs. time graph for the observation wells. Note: fluxes are negative do to the way we defined our coordinates system.

