## Example 5: Vertical Transient Unsaturated Flow in a Homogeneous Media

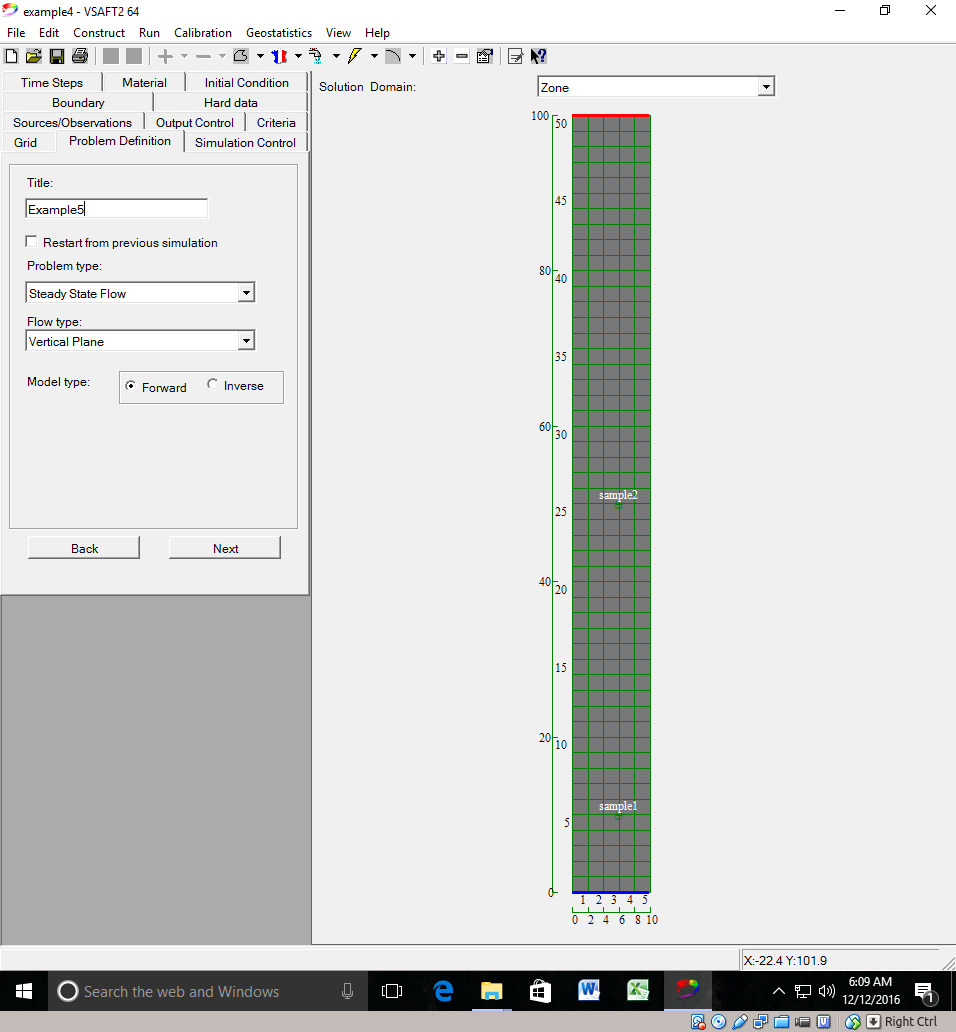
This example alters existing files from example 4. Completion of example 4 is required for completion of this example. The instructions for this example are provided as a list of steps with accompanying screen captures.

1. Open Project

* Open example 4 using **Open project** as demonstrated in example 2. Click, **Next** to go to the “Problem Definition” tab.

1. Problem Definition

* Enter a title (i.e. Example5) in the “TITLE” box. This is for record keeping purposes and to assist in remembering the details of the model. Use a descriptive title.
* Select **Transient Flow**
* Select **Next** to continue to the simulation control tab.



1. Simulation Control

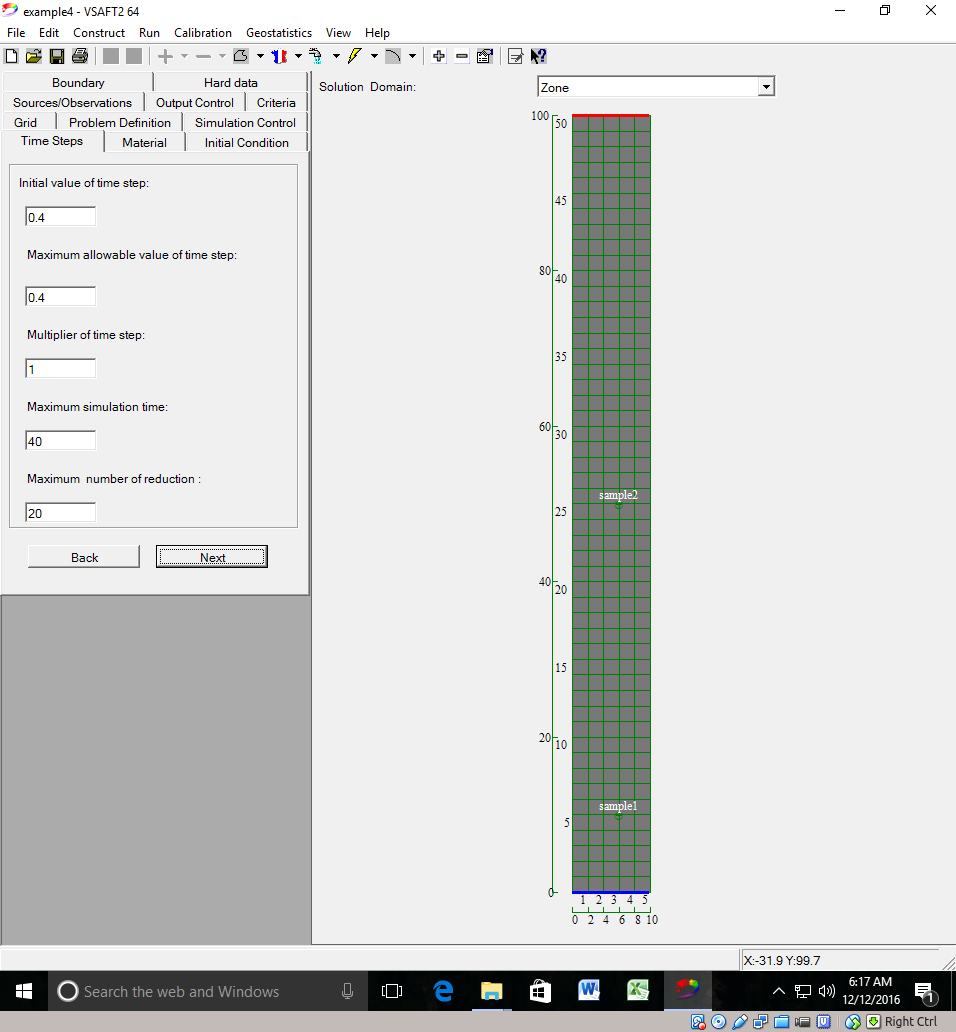
* We will accept the default simulation control parameters for this example
* Select **Next** to continue to the “time steps” tab.

1. Time Steps

In this step we will set the time step parameters do to our selection of a transient state problem.

* “Initial time step” **= 0.4**
* “Maximum time step” **= 0.4**
* “Multiplier of time step” **= 1**
* “Maximum simulation time” **= 40**
* “Maximum number of reductions” **= 20**

Select **NEXT** to continue to the “materials” tab.

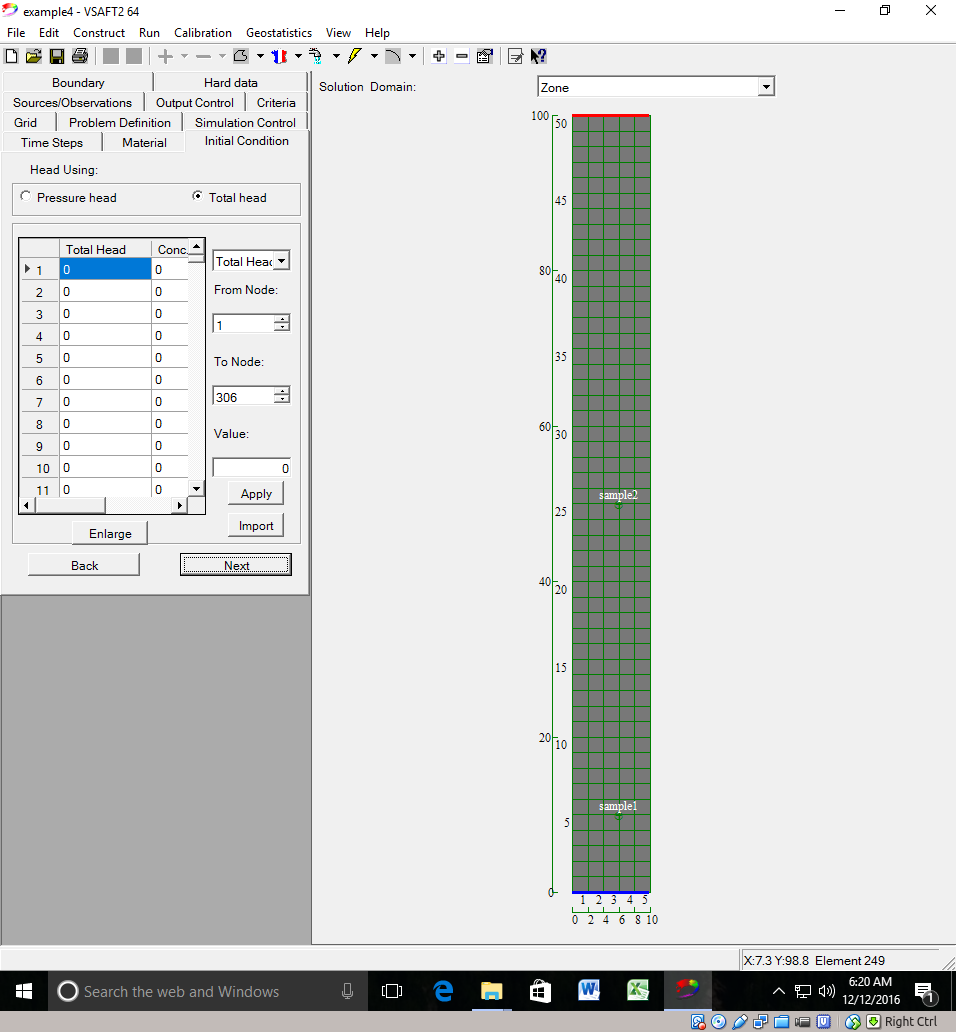


1. Materials

* The material distribution is the same as for example 4 so it does not need editing.
* Select **Next** to continue to the “initial condition” tab.

1. Initial conditions

* The initial conditions are unchanged from example 4.
* Select **Next** to continue to the “source” tab.



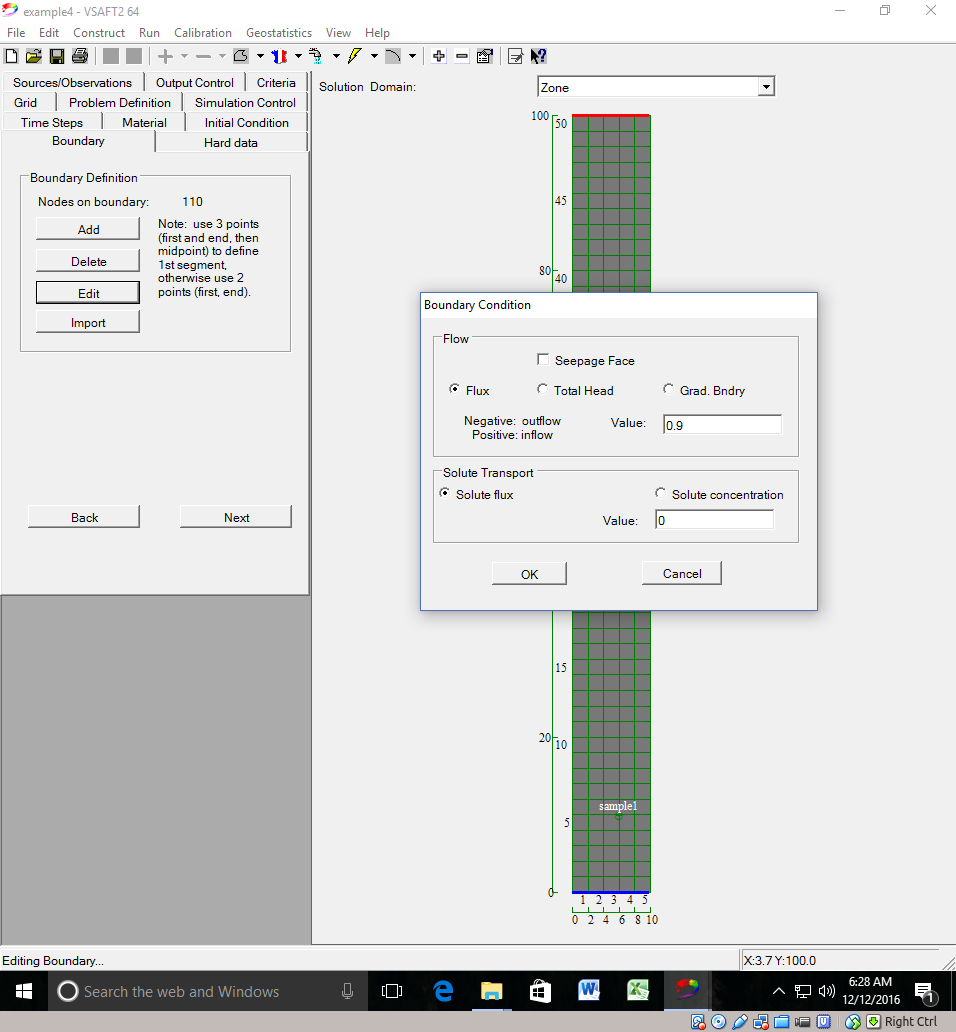
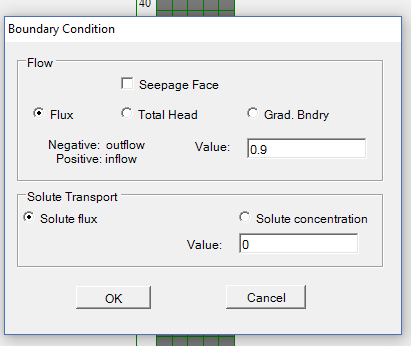
1. Sources:

* Source conditions are to be kept same as in example 4.
* Select **Next** to continue to the “boundary” tab.

1. Boundary

Edit the boundaries to reflect unsaturated flow.

* Select **Edit** then select the upper prescribed flux boundary.
* Set the flux to **0.9**, and select **OK**
* Select **Next** to continue to the “output control” tab.



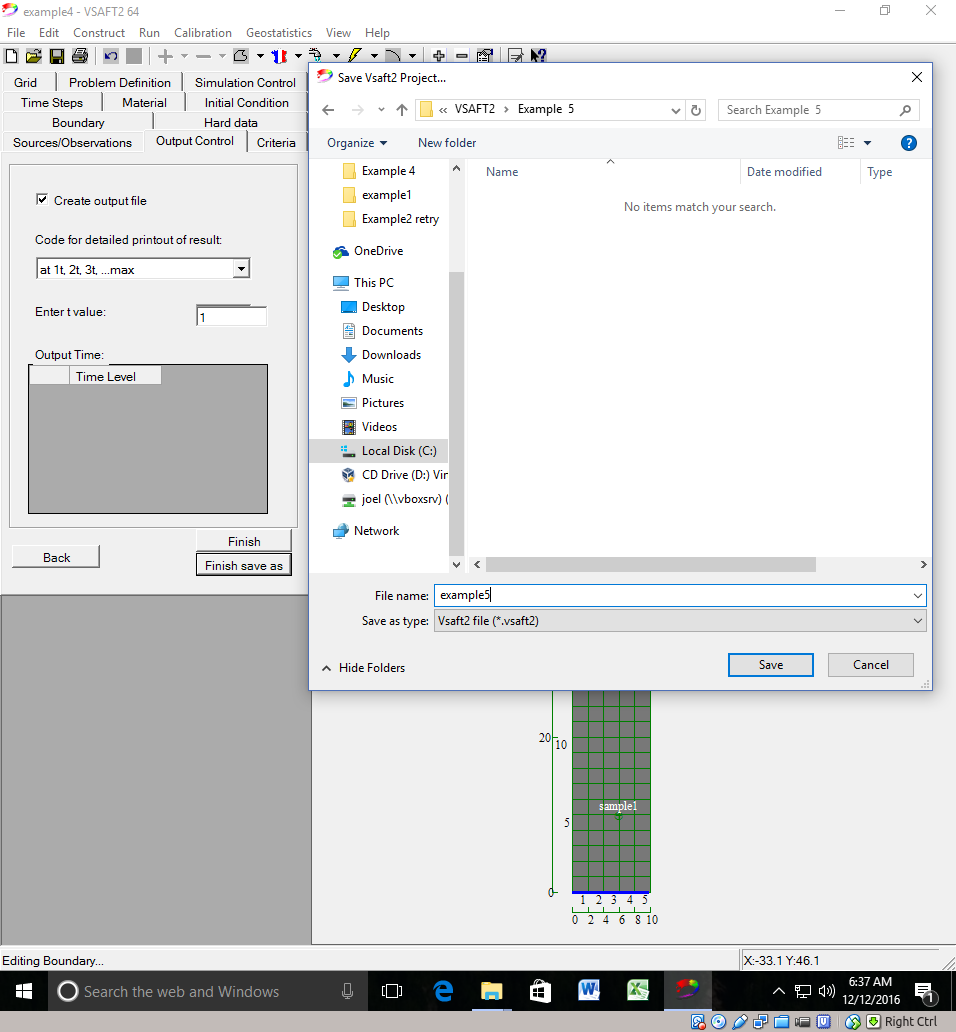
1. Sources/Observations

* The two observations wells are the same as in example three.
* Click **Next** to move to the “output control” tab.

1. Output Control

* Output control is the same as example 2.
* Select **Finish save as** to Finish the model setup and save the project to a directory.

`

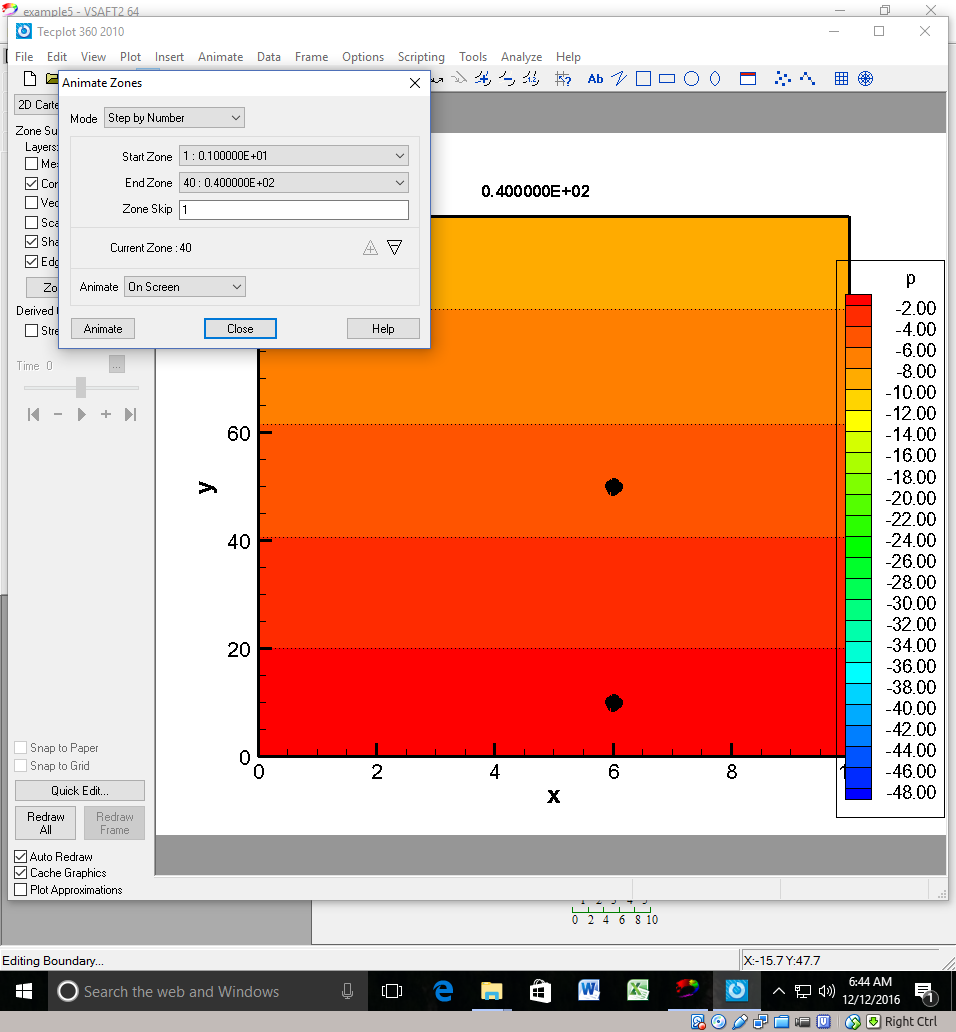


1. Running VASFT2

* Run VSAFT2 as shown in example1.

1. Viewing results in TECPLOT

* View the hydraulic head distributions using TECPLOT as explained in example 2.



1. Viewing hydrograph

* View hydraulic head change with time for the two observation points as shown in example 2.

