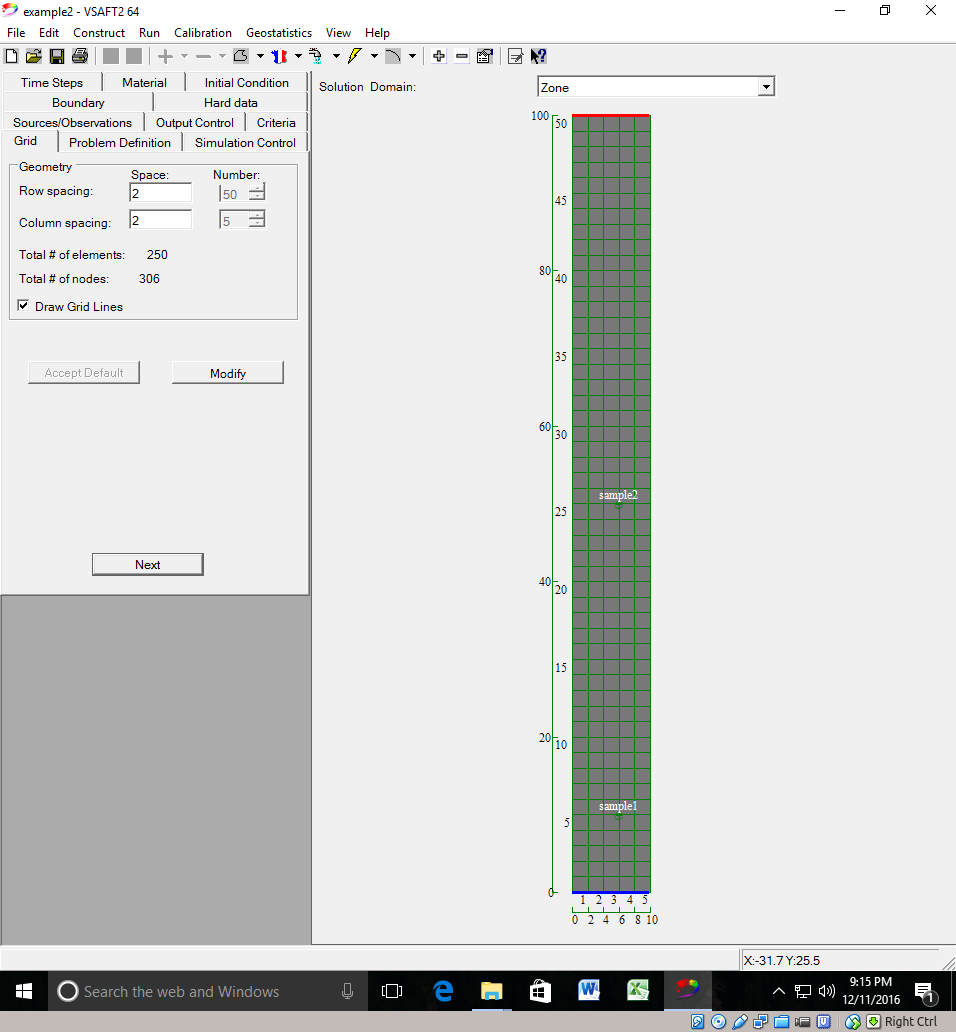
**Example 4: Vertical Steady State Unsaturated Flow in a Homogeneous Media**

In this example we will alter existing files from example 2. The instructions for this example are provided as a list of steps with accompanying screen captures.

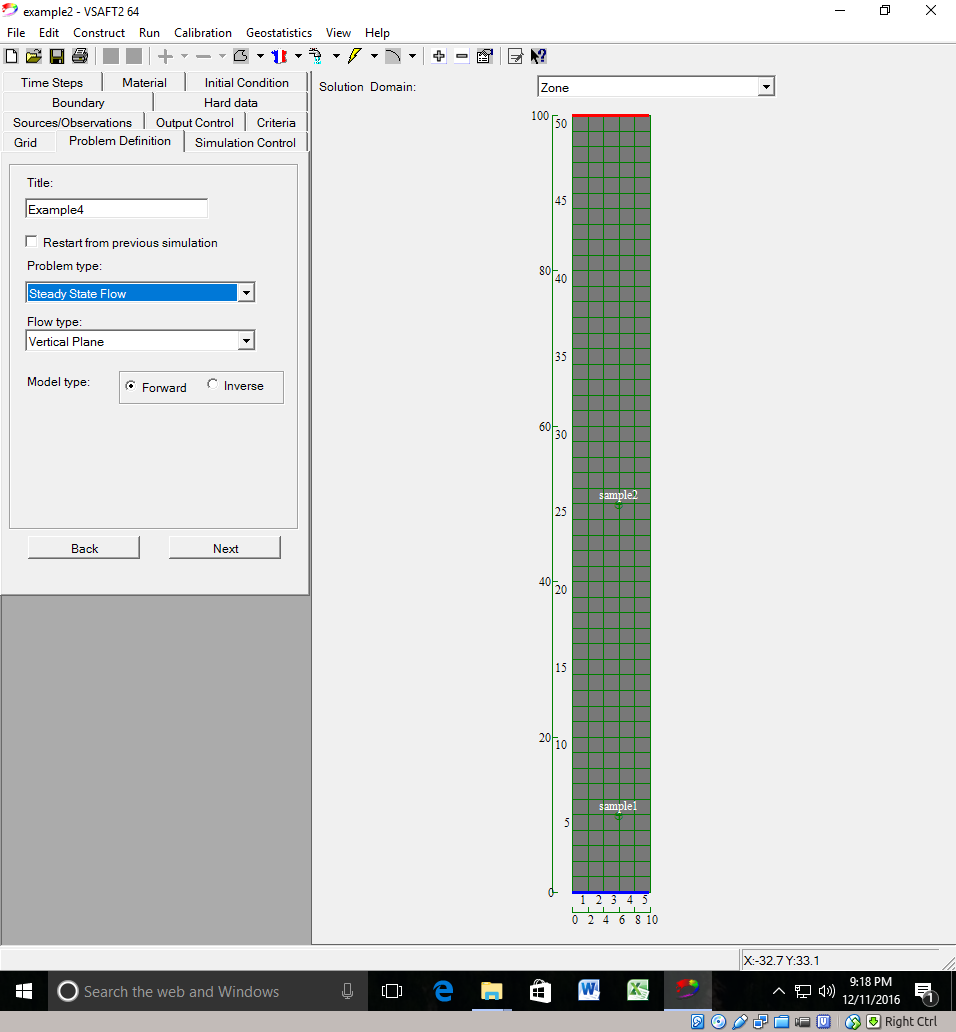
Open Project

* Open example 2 using **File->Open project** as demonstrated in example 2.
* You should have been advanced to the main VSAFT2 window. Your screen should look like the one on the below.
* Select **Next** toadvance to the “problem definition” tab.



1. Problem Definition.

* Enter a title (i.e. Example4) in the “TITLE” box. This is for record keeping purposes and to assist in remembering the details of the model. Use a descriptive title.
* Set the “problem type” to **Steady State Flow.**
* Select **Next** to continue to the “simulation control” tab.



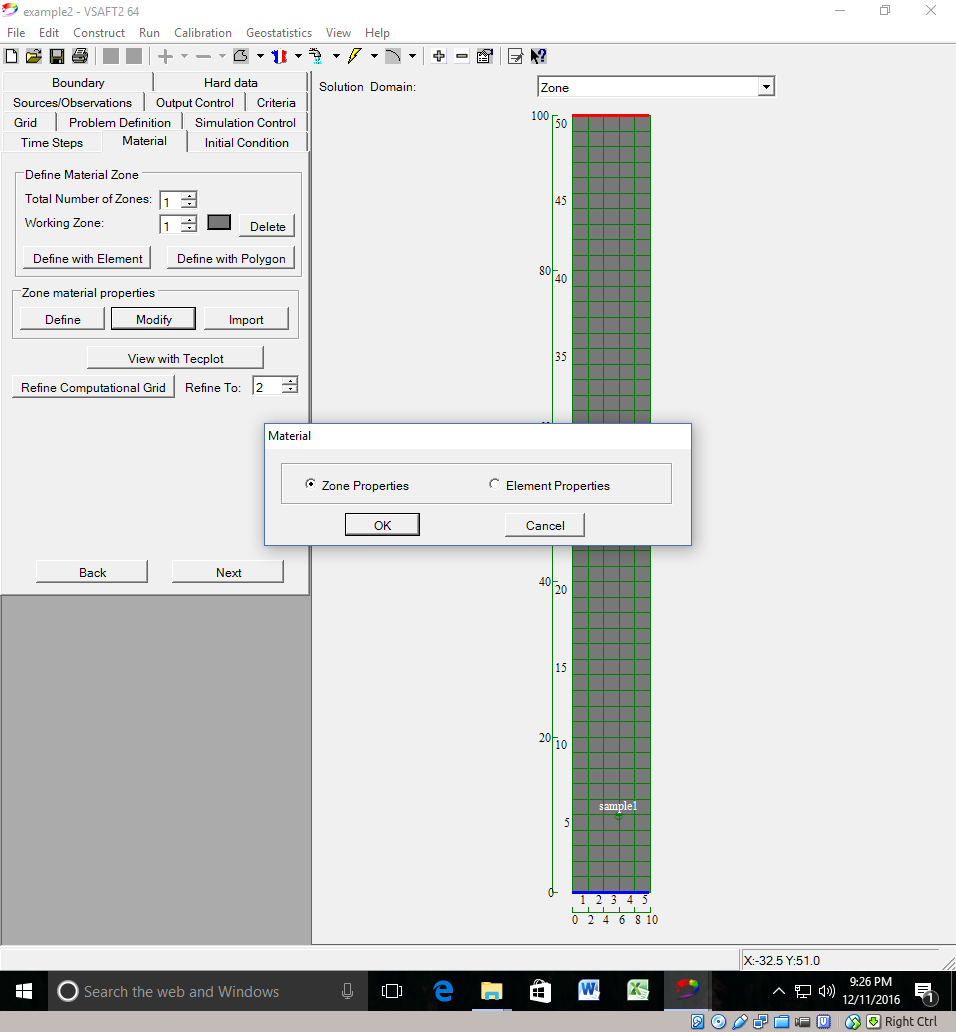
1. Simulation Control

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

1. Materials

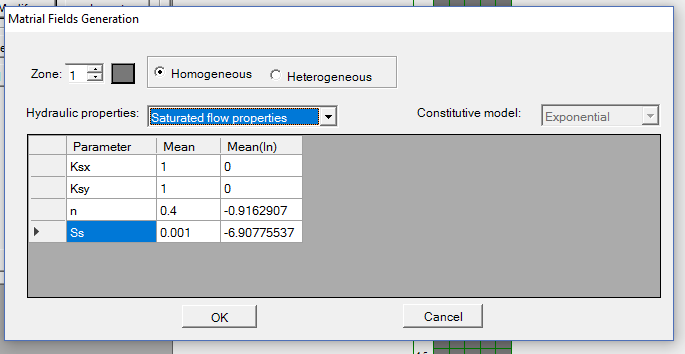
The material distribution is the same as for example 2 so it does not need editing.

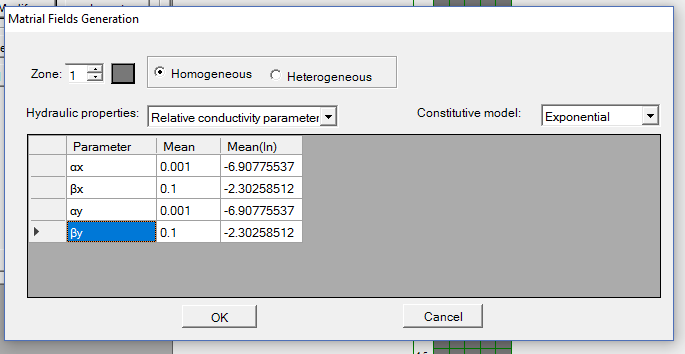
* Select **Modify,** to define the material as a zone. Select the **Zone properties** in pop up window then select **OK**.

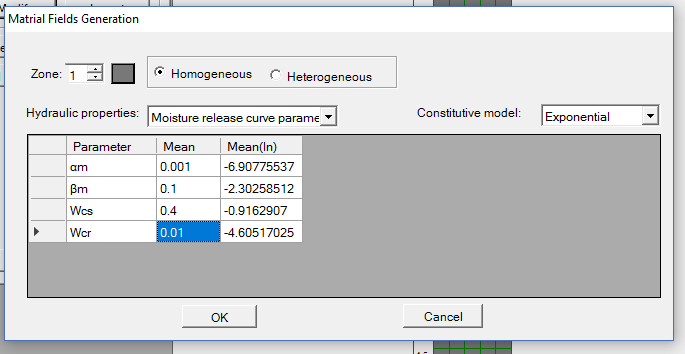


Enter the following in the “Material Fields Generation” dialog box:

|  |  |  |
| --- | --- | --- |
| Enter the values for the saturated flow properties in Zone 1.  Make sure **Homogenous** is selected.   * Ksx = **1.0** * Ksy = **1.0** * n = **0.4** * Ss = **0.001** | Select **Relative conductivity parameters** from the hydraulic properties menu.  Enter the values for the relative conductivity parameters in Zone 1.  **αx =0.001**  **βx =0.1**  **αy =0.001**  **βy =0.1**  Select **Exponential** constitutive model | Select the unsaturated **moisture release curve** from the hydraulic properties menu.  Enter the values for the relative conductivity parameters in Zone 1.  **αm =0.001**  **βm =0.1**  **Wcs =0.4**  **Wcr =0.01**  Select **Exponential** constitutive model |



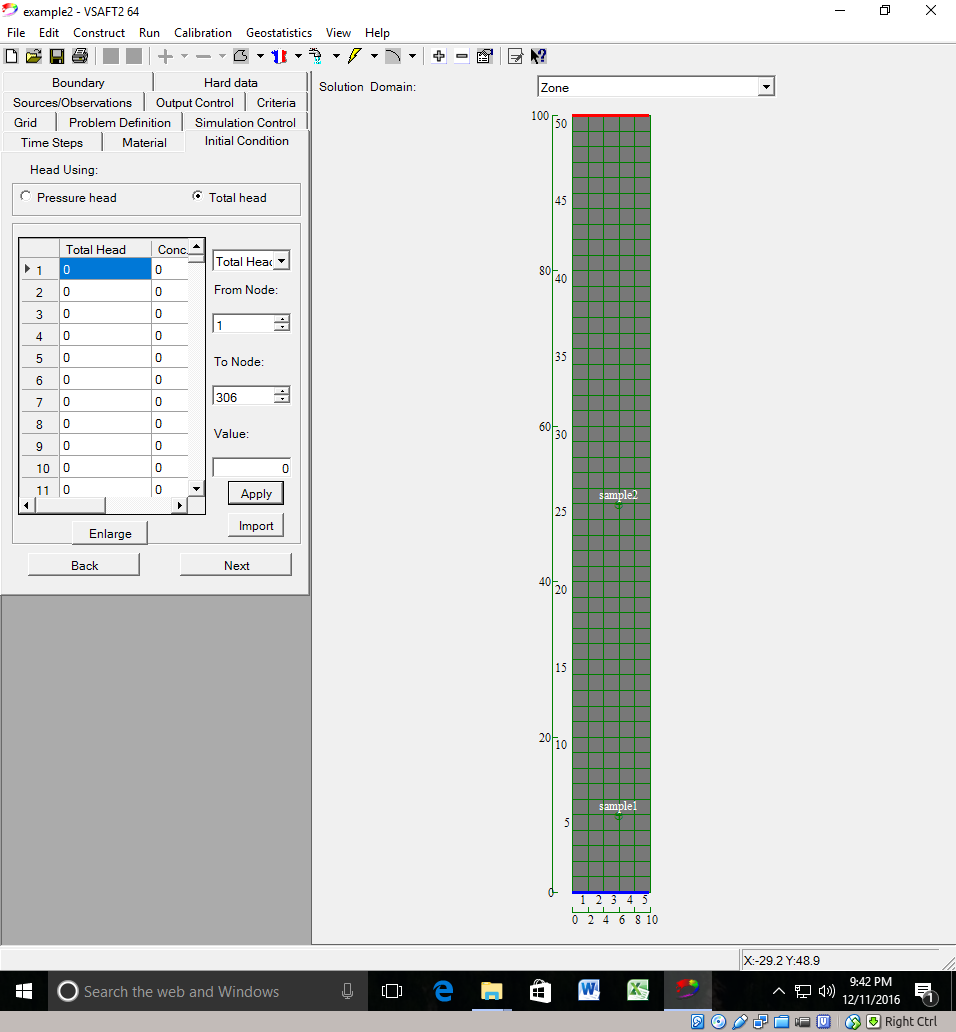




* Once all parameters are imputed into the “Material Field Generation” select **OK**.
* Select **Next** to continue to the “initial condition” tab.

1. Initial conditions

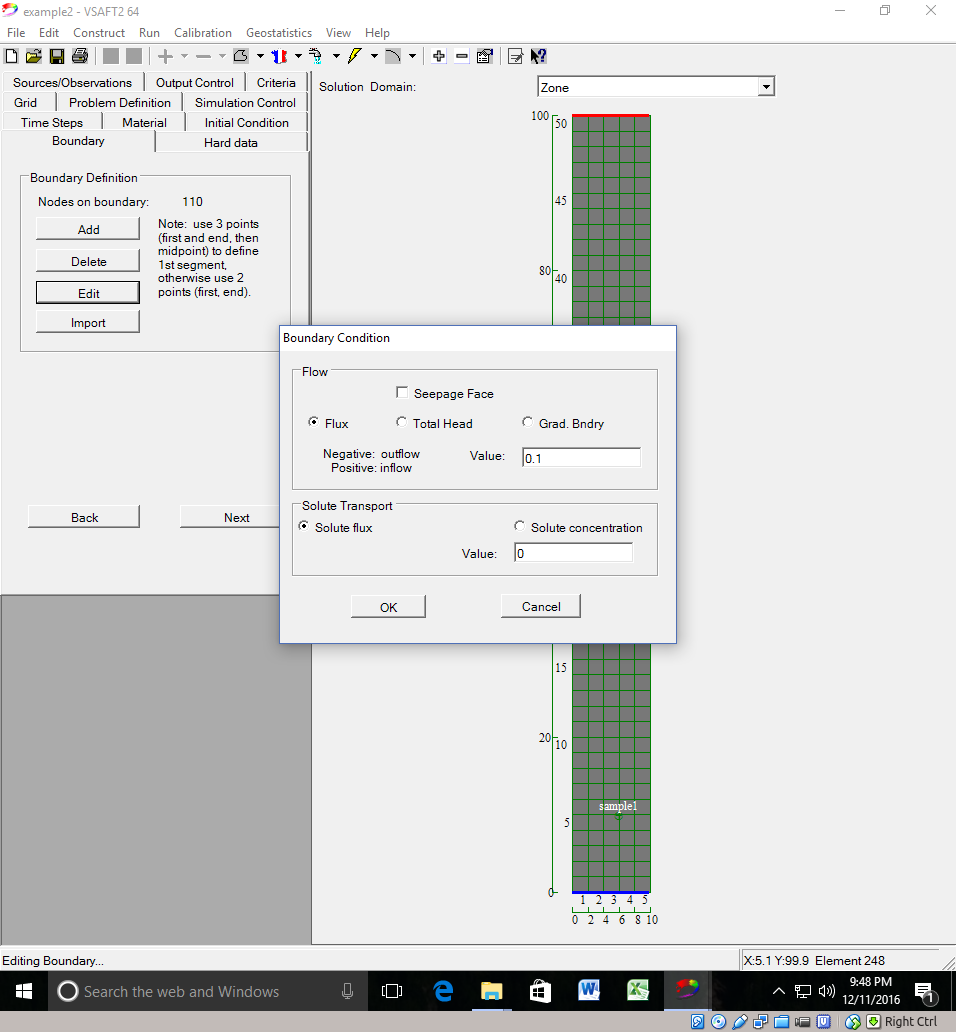
* To set the initial pressure head conditions, select **Total head**
* Enter **0 cm** in the “Value” box and select **Apply.**
* Select **Next** to continue to the “boundary” tab.



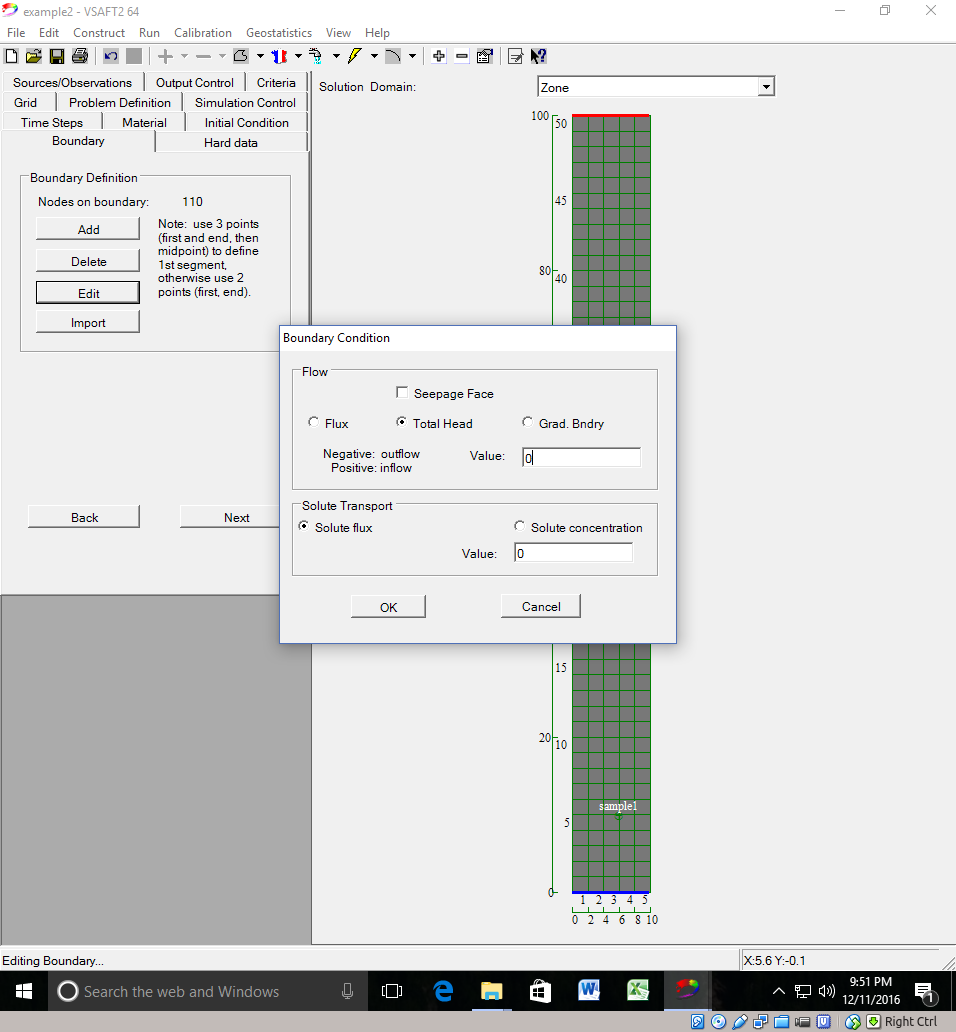
1. Boundary

Edit the boundaries to reflect unsaturated flow.

* Select **Edit,** then select the upper prescribed flux boundary.
* Set the flux to **0.1**, and select **OK**



* Select **Edit** then select the lower prescribed pressure head boundary.
* Set the pressure head to **0**, and select **OK**
* Select **Next** to continue to the “Sources/observations”.

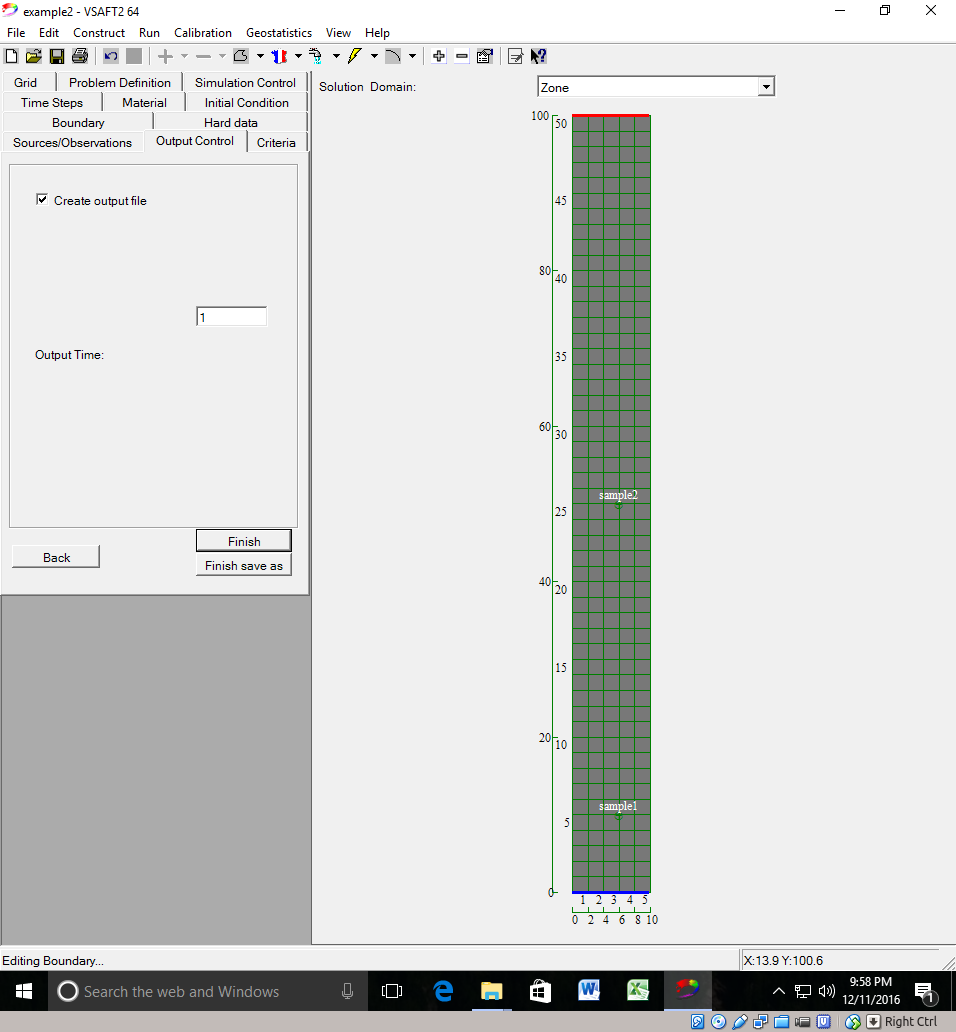


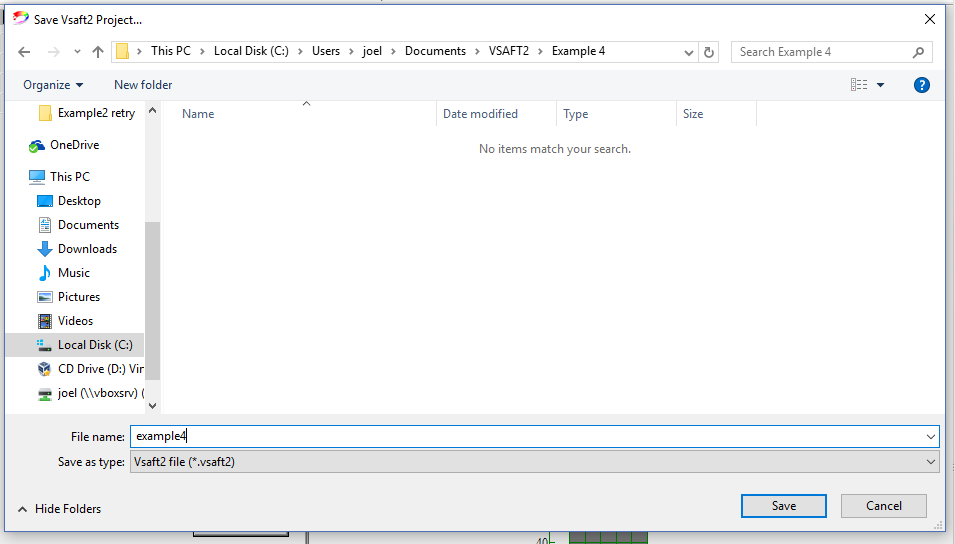
1. Sources/observations

* Sources and observations are same as in example 2 so no need to edit them.
* Select **Next** to continue to the “output control” tab.

1. Output Control

* Settings in the “Output control” tab are the same as example 2.
* Select **Finish save as** to finish the model setup and save the project to a directory.
* Save your project with the name **example4** and select **OK**
* The model is now defined and the input files for VSAFT have been generated.

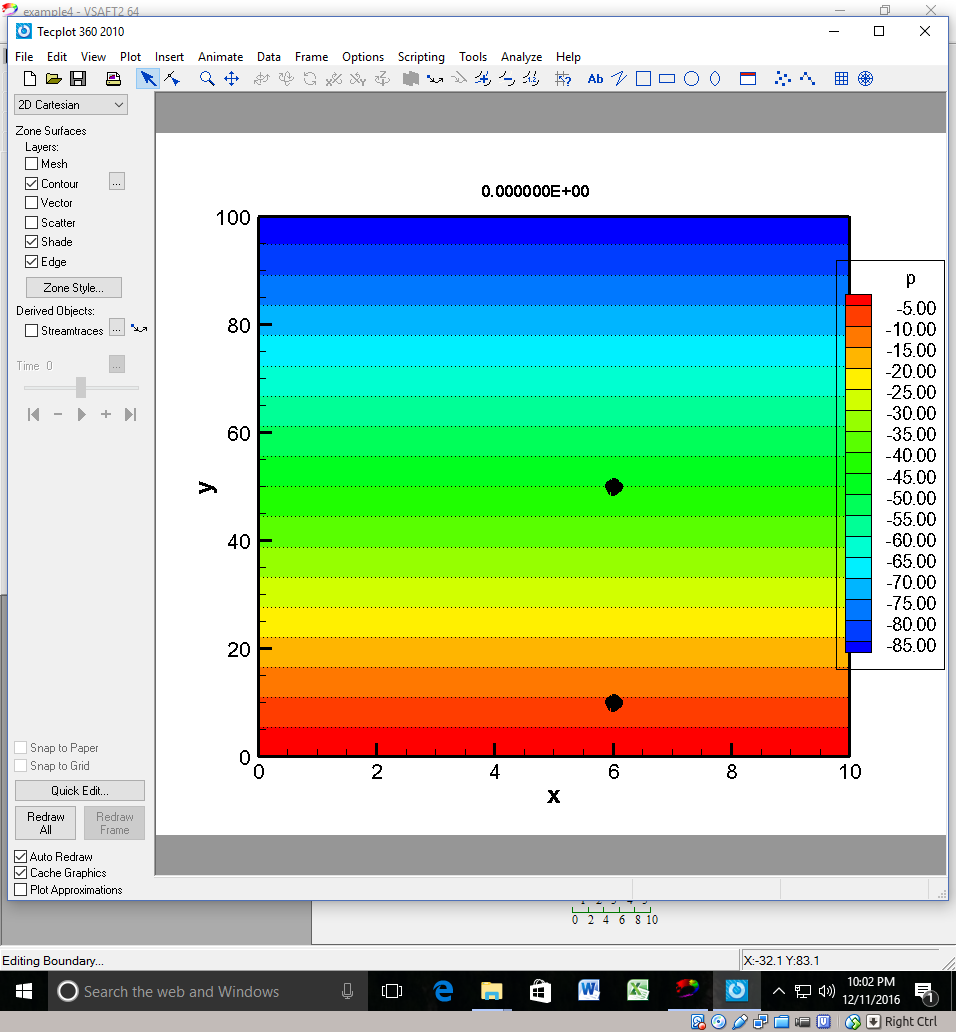




1. Running VASFT2

* Run VSAFT2 as shown in example1.

1. Viewing results in TECPLOT

* View the hydraulic head distributions using TECPLOT as explained in example 1.