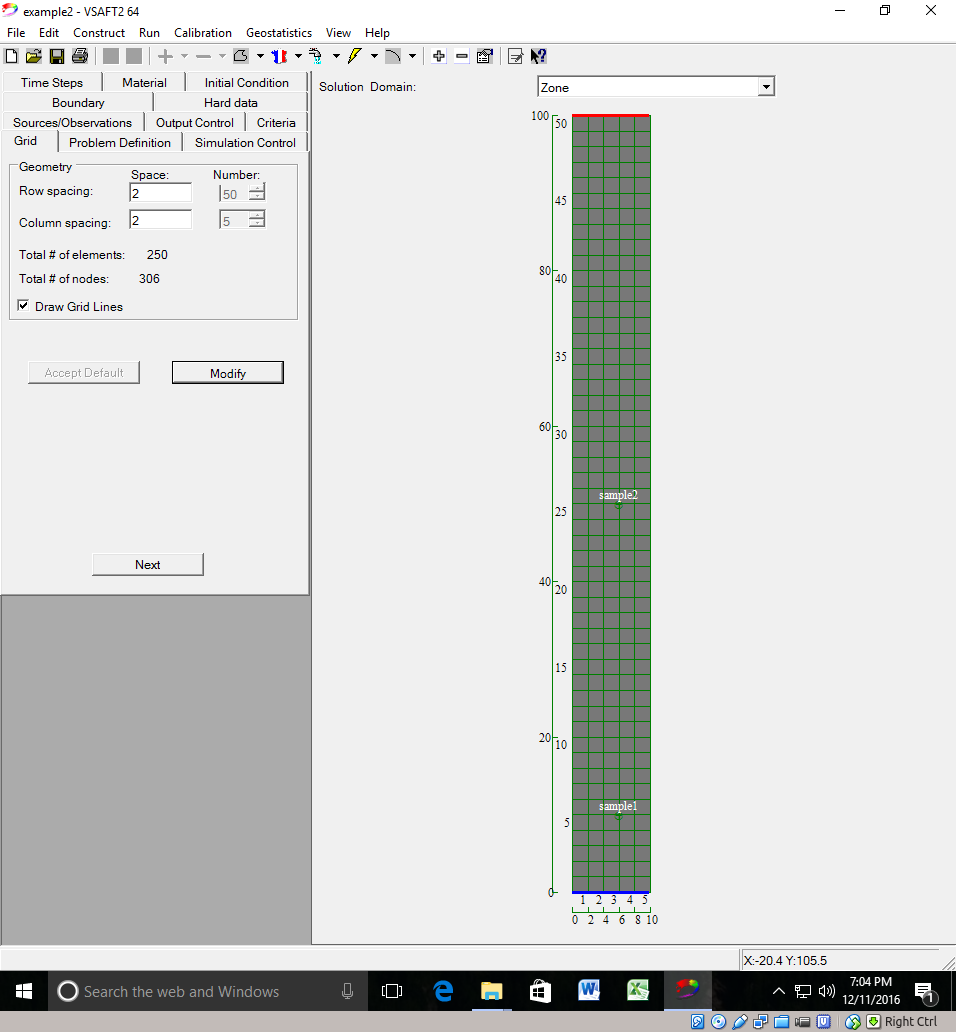
**Example 3: Vertical Transient Flow in a Two Layer Media**

The instructions for this example are provided as a list of steps with accompanying screen captures. The example builds on the model generated in example 2. To complete this example you need to have completed example 2.

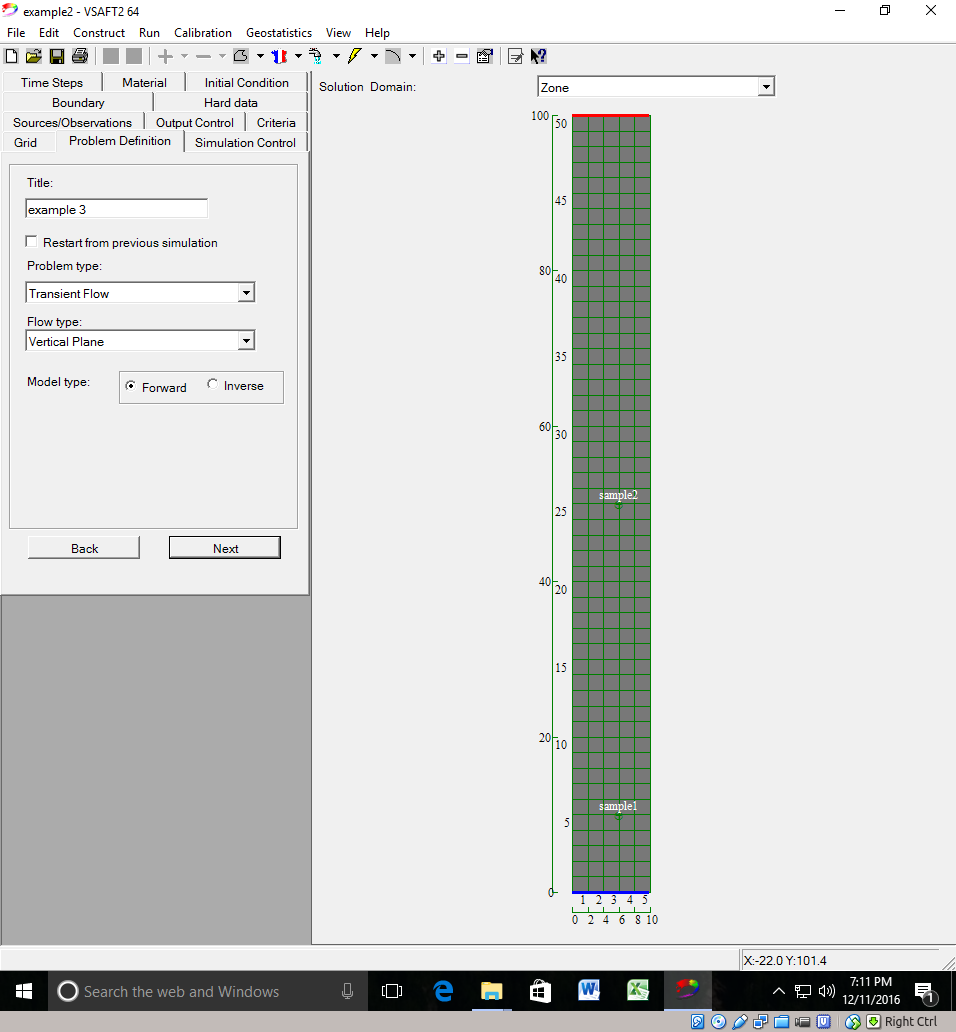
1. Open Project

* Open example 2 using Open project as demonstrated in example 2
* You should have been advanced to the main VSAFT2 window and the tab Grid should be selected Your screen should look like the one below.
* Select Next to advance to the problem definition tab.



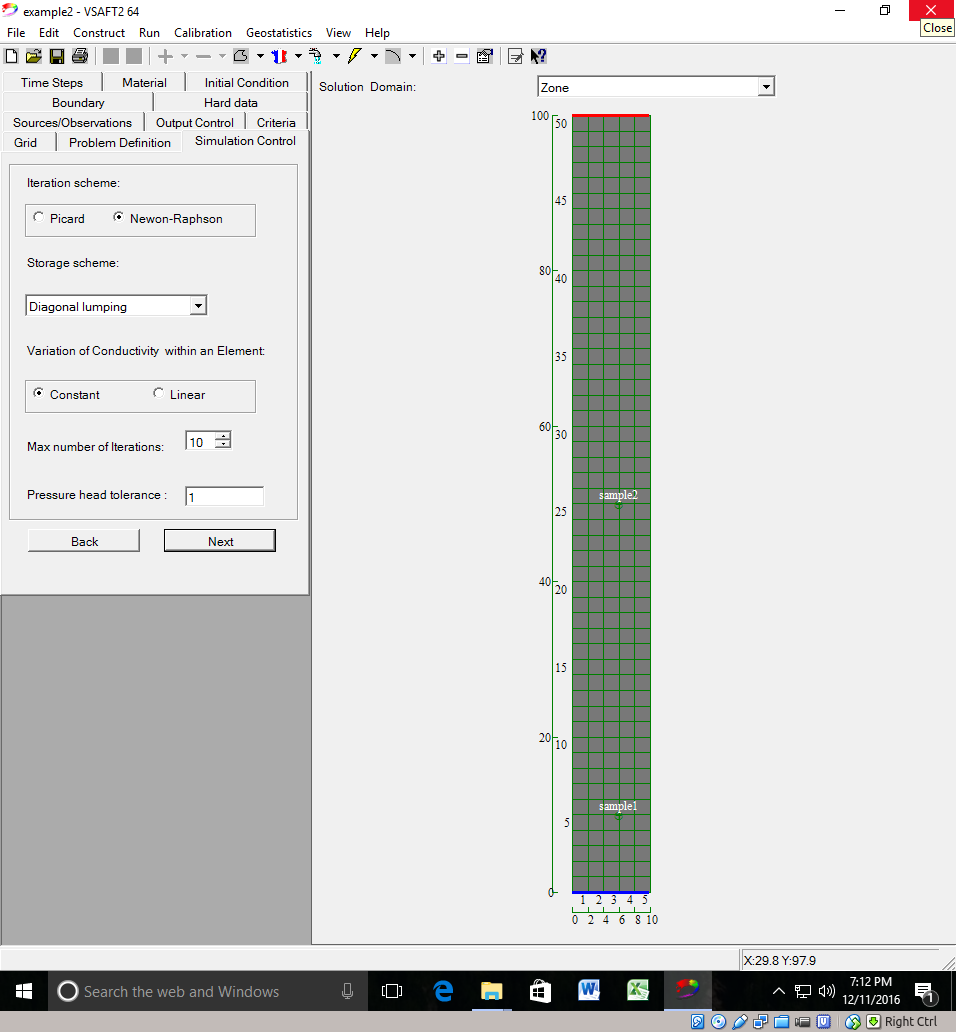
1. Problem Definition.

* Enter a title (i.e. Example3) 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 **Next** to continue to the “simulation control” tab.



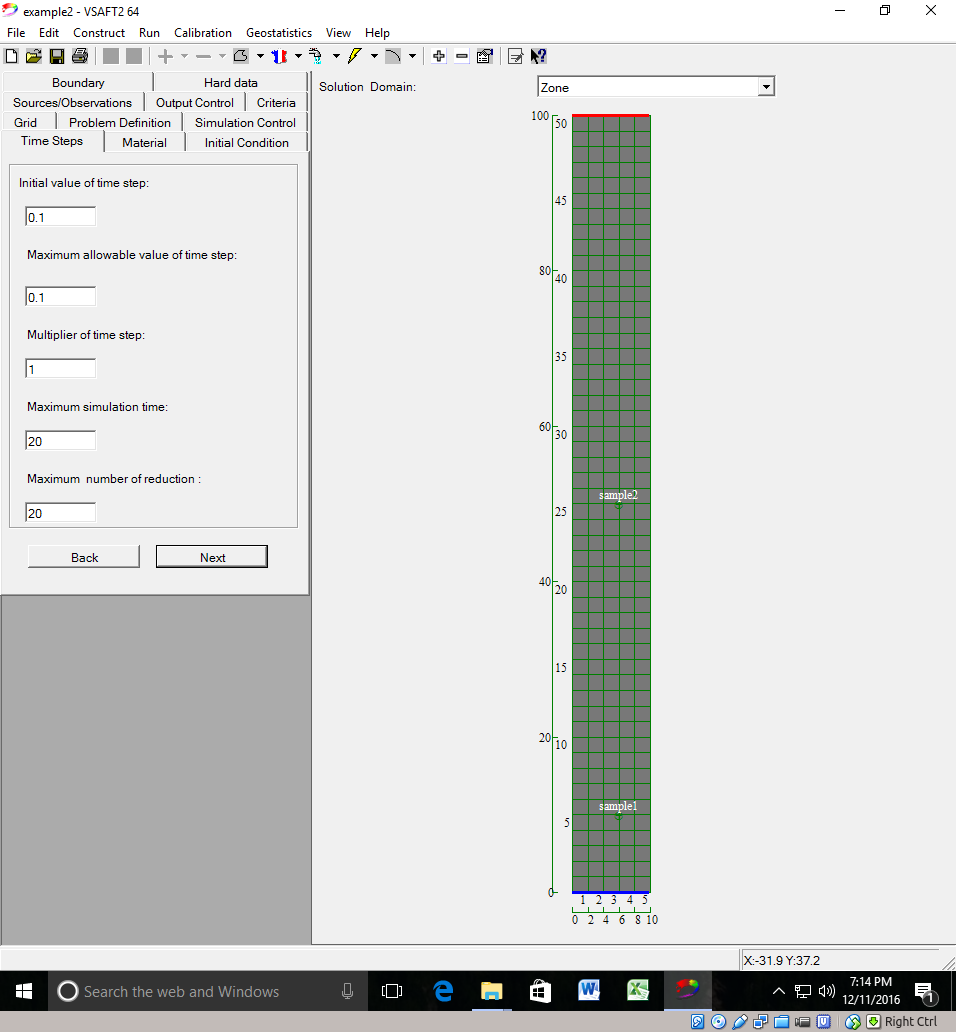
1. Simulation Control

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



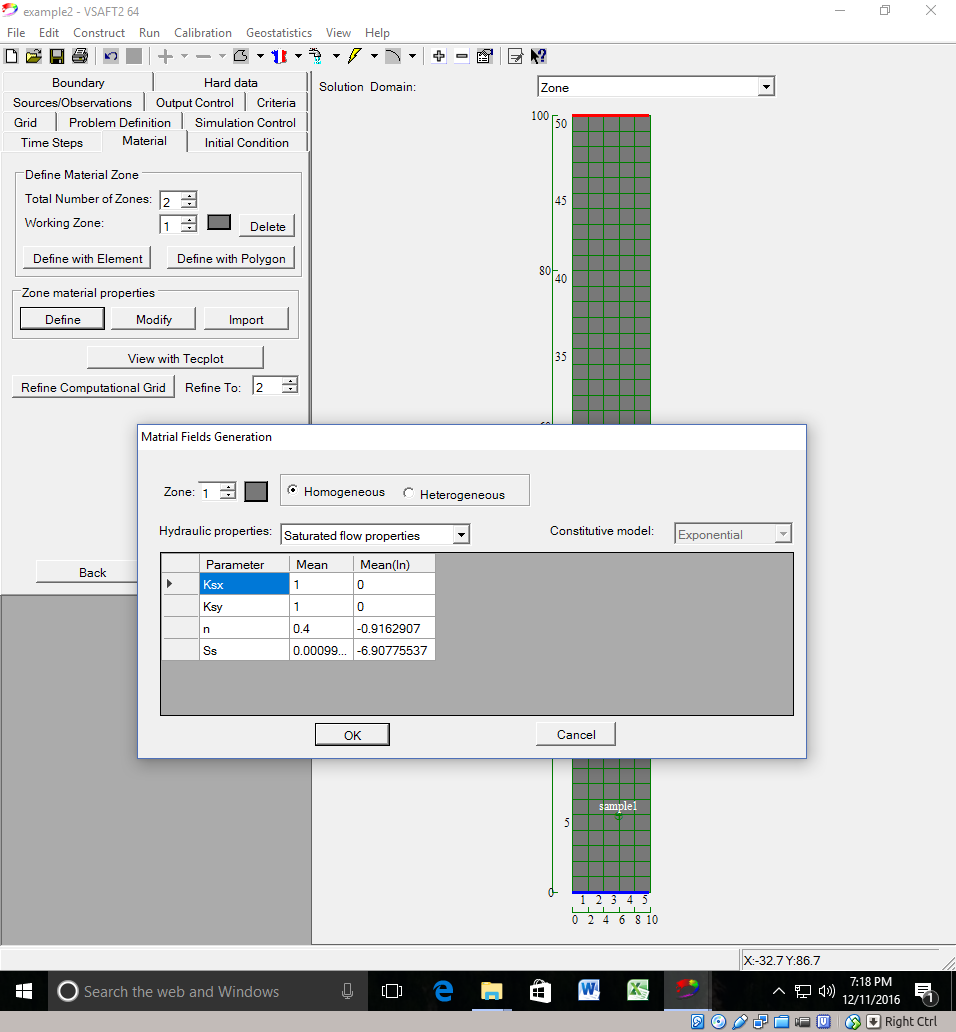
1. Time Steps

* Use the same time step parameters as in example2.
* Select **NEXT** to continue to the “materials” tab.



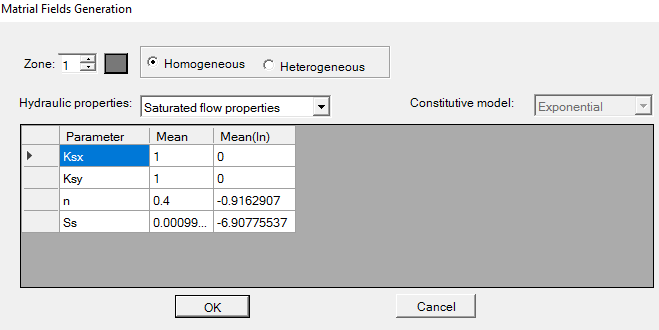
1. Materials

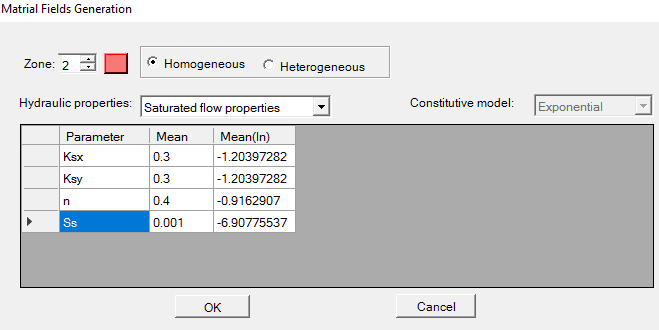
* This example uses two different materials to define the characteristics of the porous matrix.
* Change the “Total Number of Zones” in the “Define Material Zone” area to **2**.
* Select **Define** under the “Zone material properties” location. This will open the “Material Fields generator”.



Enter the following into the “Material Fields Generator” then select **OK**:

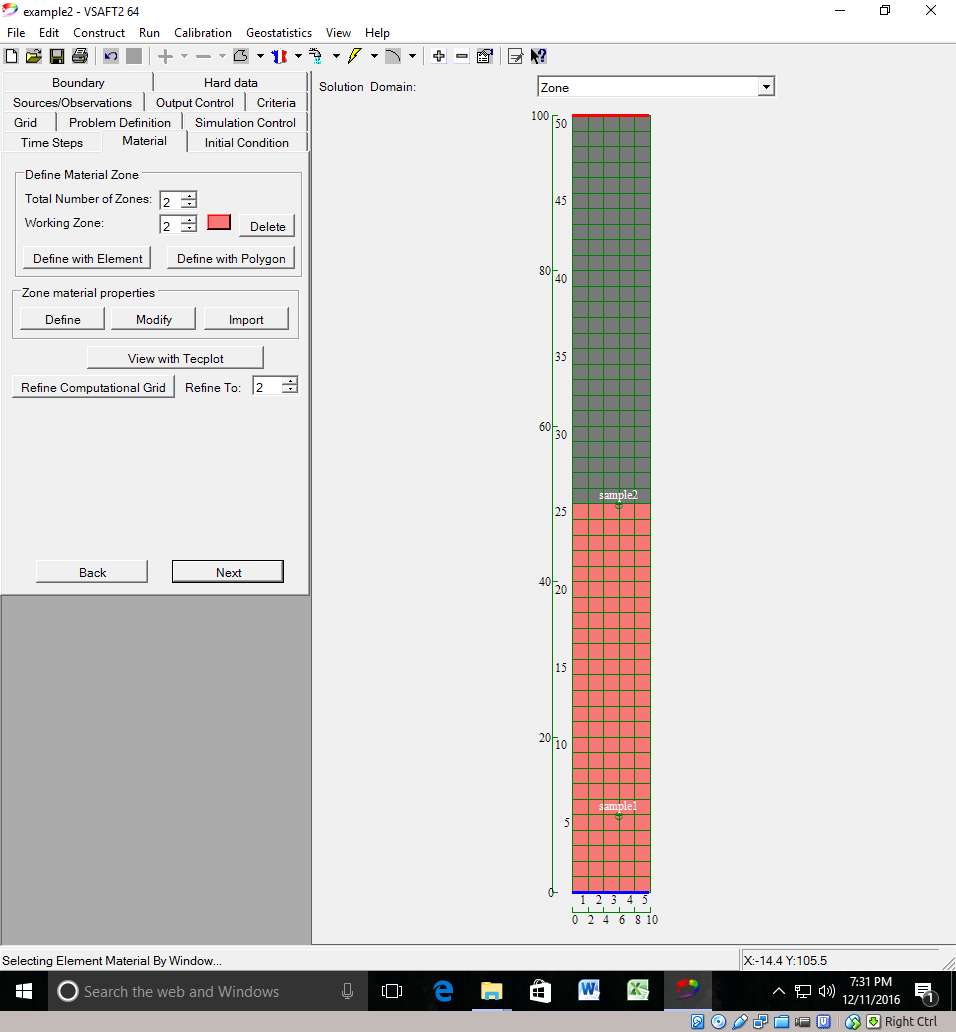
|  |  |
| --- | --- |
| For zone 1 (same as in example 2):  Select: **Homogeneous**  Select: **Saturated flow properties**  Ksx = **1**  Ksy = **1**  n = **0.4**  Ss = **0.001** | For zone 2:  Select: **Homogeneous**  Select: **Saturated flow properties**  Ksx = **0.3**  Ksy = **0.3**  n = **0.4**  Ss = **0.001** |





Next, you need to define the material zones using either “Define with Elements” or “Define with Polygon” buttons. In this case we will define the two material zones with “Define with Polygon”.

* All elements of the grid are, by default, set to zone 1. Here we will set the blocks that will be zone 2, making the system heterogeneous.
* Set the “working zone” to **2** to define material 2 as the element you want to define in your domain.
* Select **Define with Polygon**.
* Using the mouse define the bottom half of the column as material 2 (zone 2).
* Select **Next** to continue to the “initial condition” tab.



1. Initial conditions

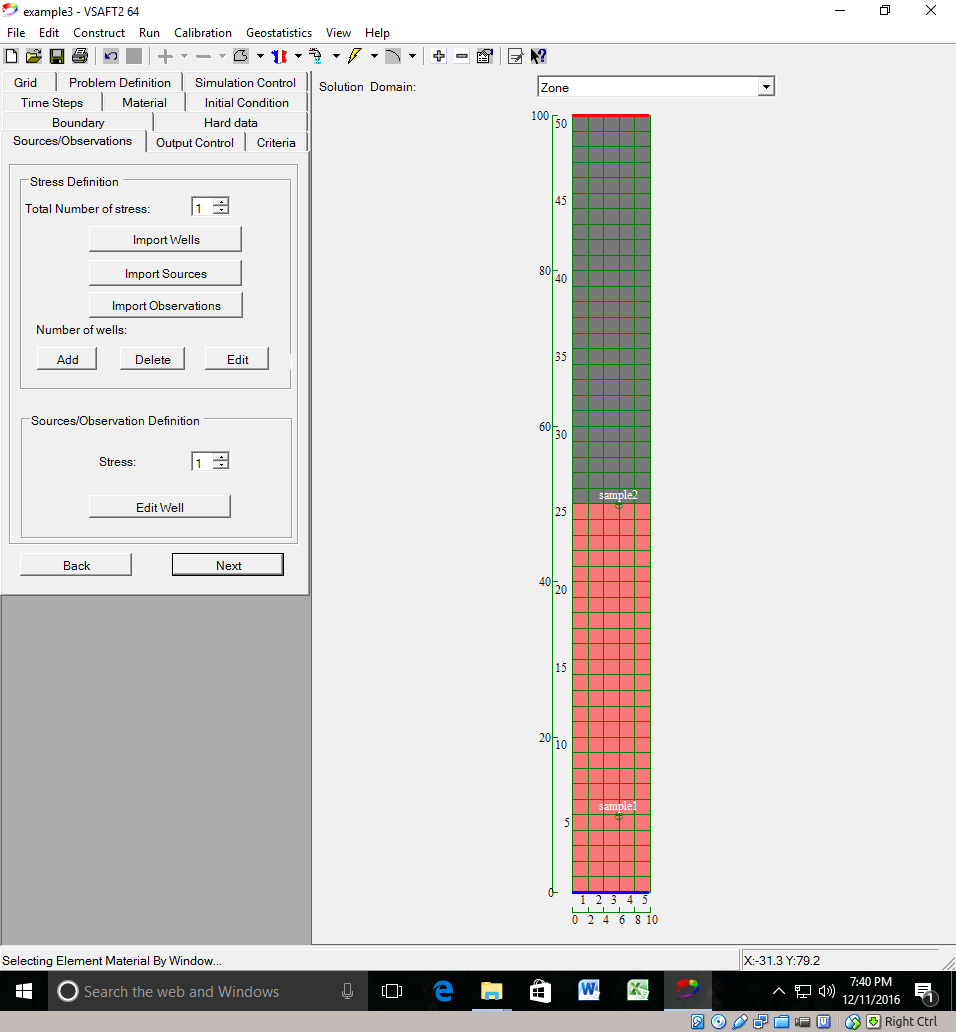
* The initial conditions are the same as for example 2 so they do not need editing.
* Select **Next** to continue to the “Boundary” tab.

1. Boundary

* The boundary conditions are the same as example 2 so they do not need editing
* Select **Next** to continue to the “Source” tab.

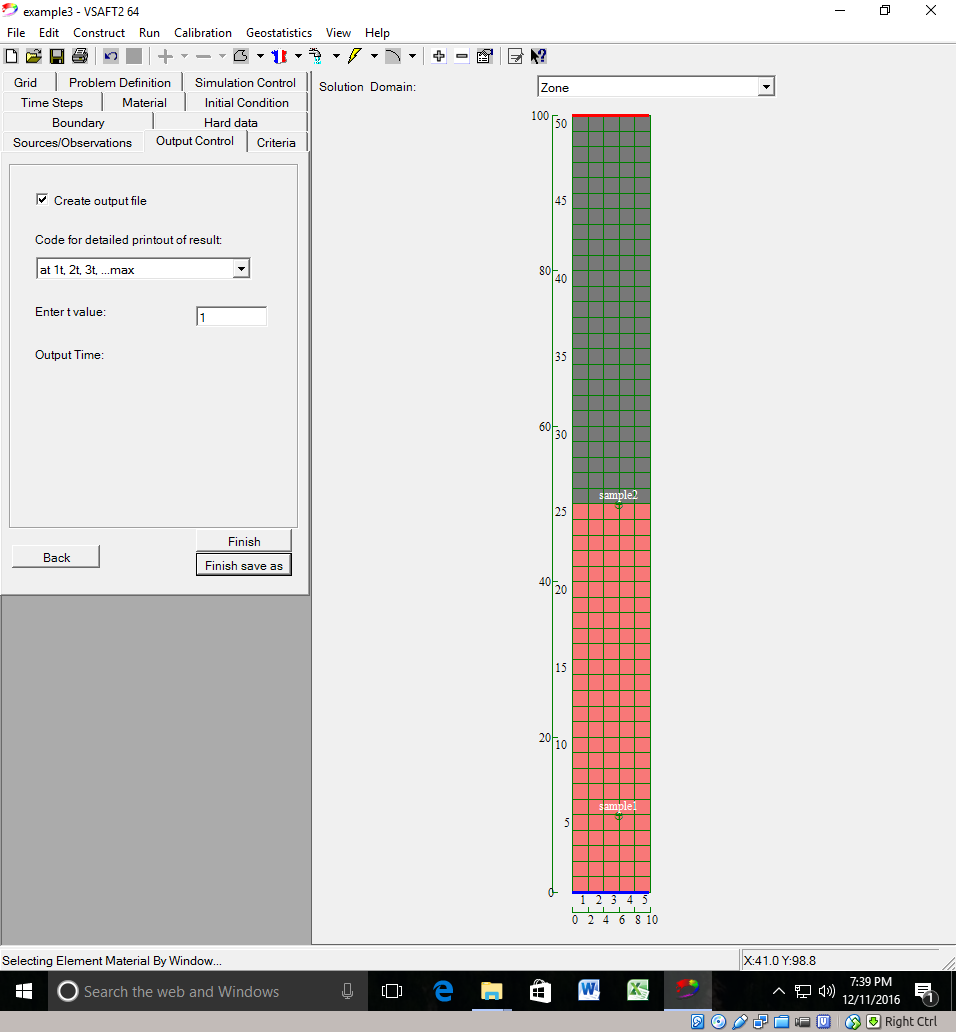
1. Source:

* The observation wells are the same wells defined in example 2 so they do not need editing
* Select **Next** to continue to the “output control” tab.



1. Output Control

* Use the same output control settings 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 **example3** and select **OK**. Instructions for saving are included in example 2.



1. Running VASFT2

* Run VSAFT2 as shown in example1.

1. Viewing results in TECPLOT

* Run TECPLOT as shown in example 2.

The hydrograph results from Example 3.

