## Example 7: Vertical steady state flow and transport in a heterogeneous media

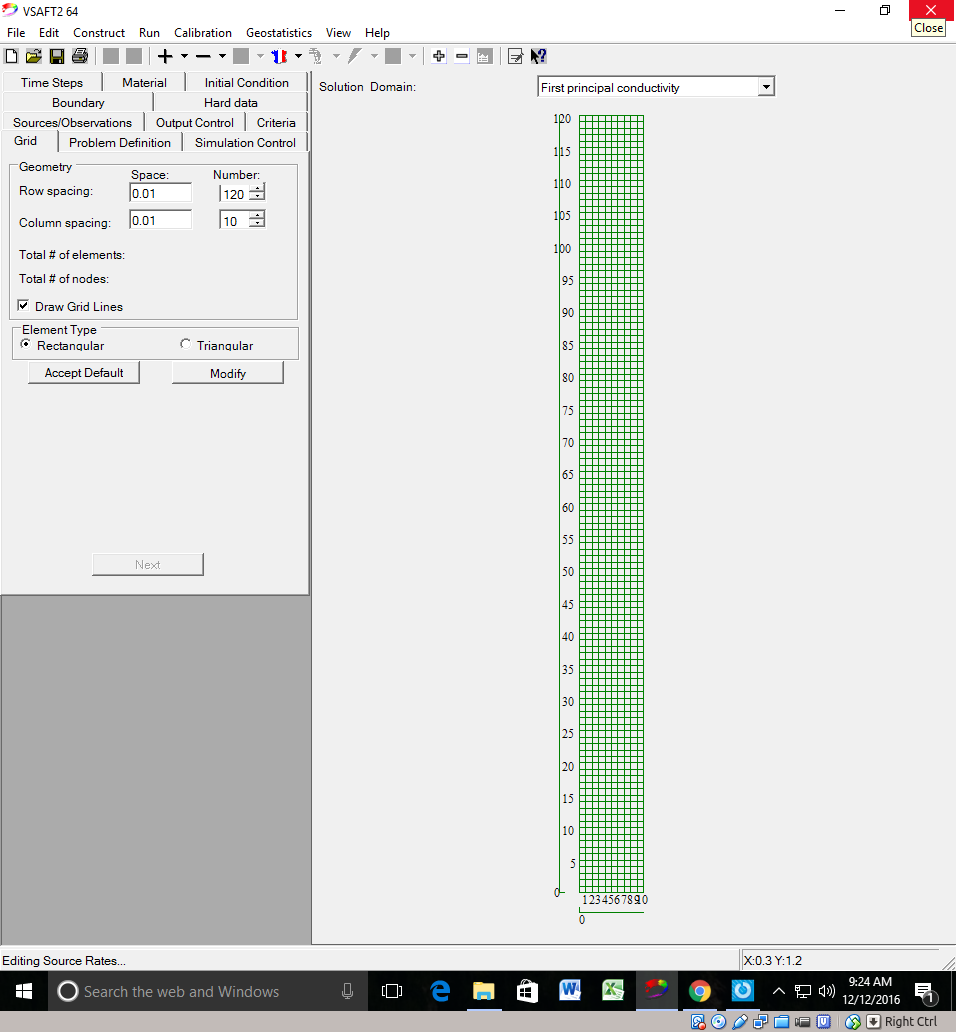
The instructions for this example are provided as a list of steps with accompanying screen captures.

1. New Project

Start a new project.

* Select **File** then **New Project**
* Define the initial grid dimensions (you can edit this later).
* Change the value for the number of rows to **120.**
* Change the value for the number of columns to **10**
* Change the row and column spacing to **0.01**.
* Then select **OK**

You should have been advanced to the main VSAFT2 window. Your screen should look like the one below.

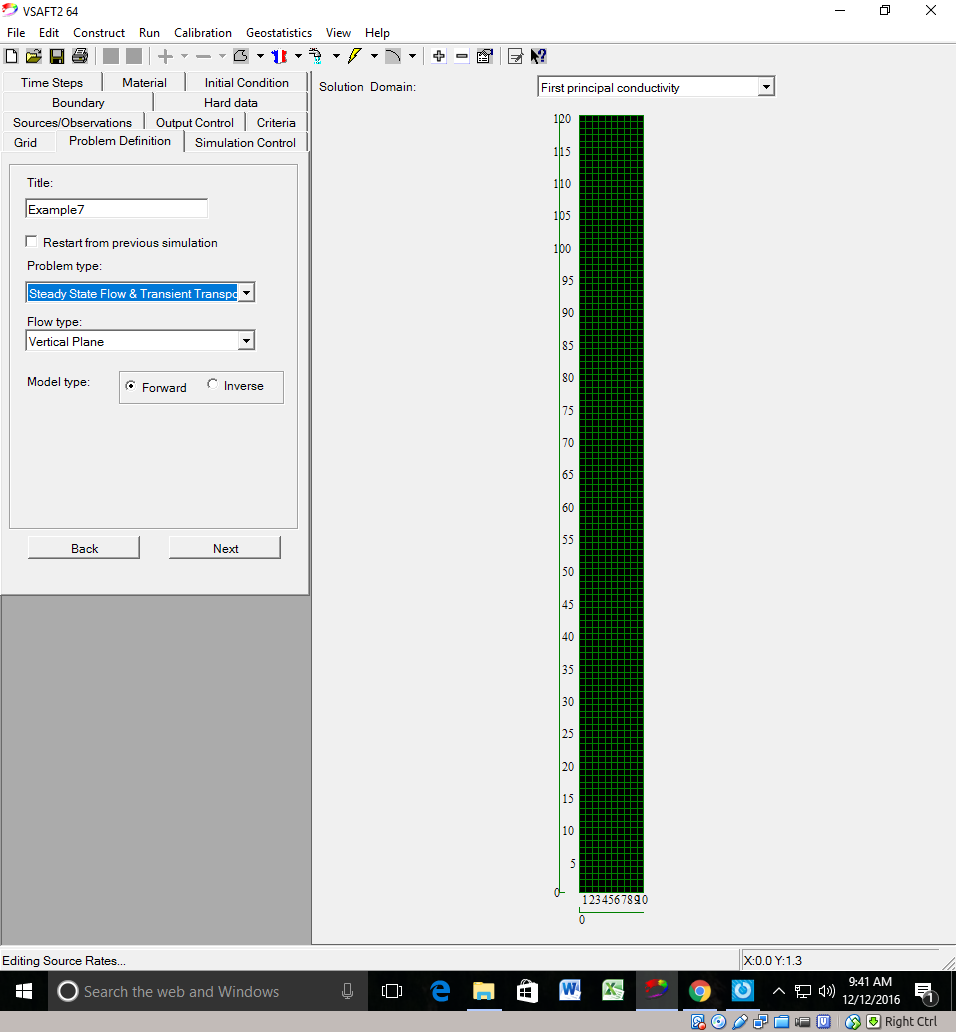


For this example we will accept the grid without editing the row or column spacing or adding additional rows or columns.

* Select **Accept Default** then you will be advanced to the “problem definition” tab.

1. Problem Definition

* Enter a title in the “TITLE” box; **Example7**. This is for record keeping purposes and to assist in remembering the details of the model.
* Use the “Problem type” drop down menu to select **Steady State Flow & Transient Transport**
* Use the “Flow type” drop-down menu to select **Vertical Plane**
* Select **Next** to continue to the “simulation control” tab.



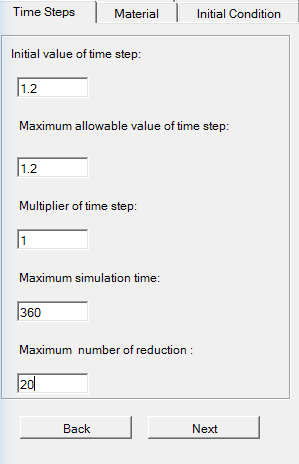
1. Simulation Control

* 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 **1.2.**
* Set the “maximum allowable value of time step” to **1.2.**
* Set the“multiplier of time step” to **1.**
* Set the “maximum simulation time” to **360.**
* Set the“maximum number of reductions” to **20**

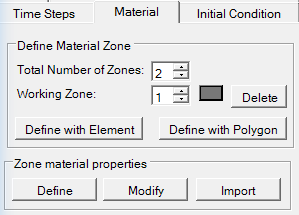


Select **NEXT** to continue to the “materials” tab.

1. Materials

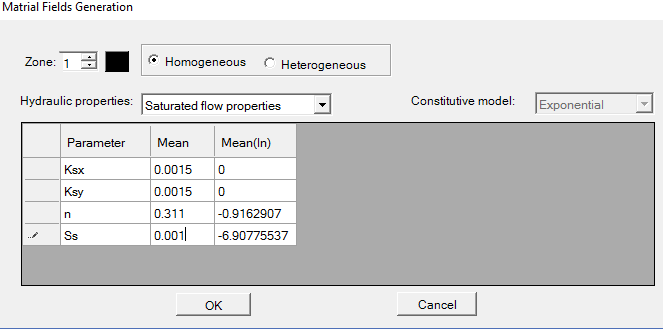
This example uses two different materials:

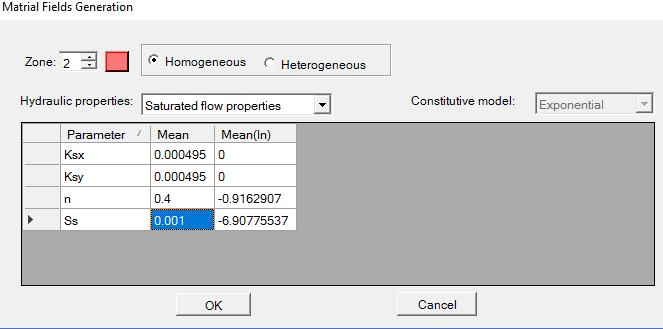
* Change the “Total Number of Zones” to **2**.
* Then under the Zone material properties select **Define.**



Enter the material properties for both zones.

|  |  |
| --- | --- |
| For Material 1:   * Change “Working Zone” to **1** * Select **saturated flow properties** * Select **Homogeneous** * Ksx = **0.0015** * Ksy = **0.0015** * n = **0.311** * Ss= **0.001** | For Material 2:   * Change “Working Zone” to **2** * Select **saturated flow properties** * Select **Homogeneous** * Ksx =**0.000495** * Ksy = **0.000495** * n= **0.4** * Ss = **0.001** |

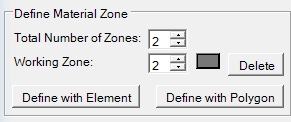




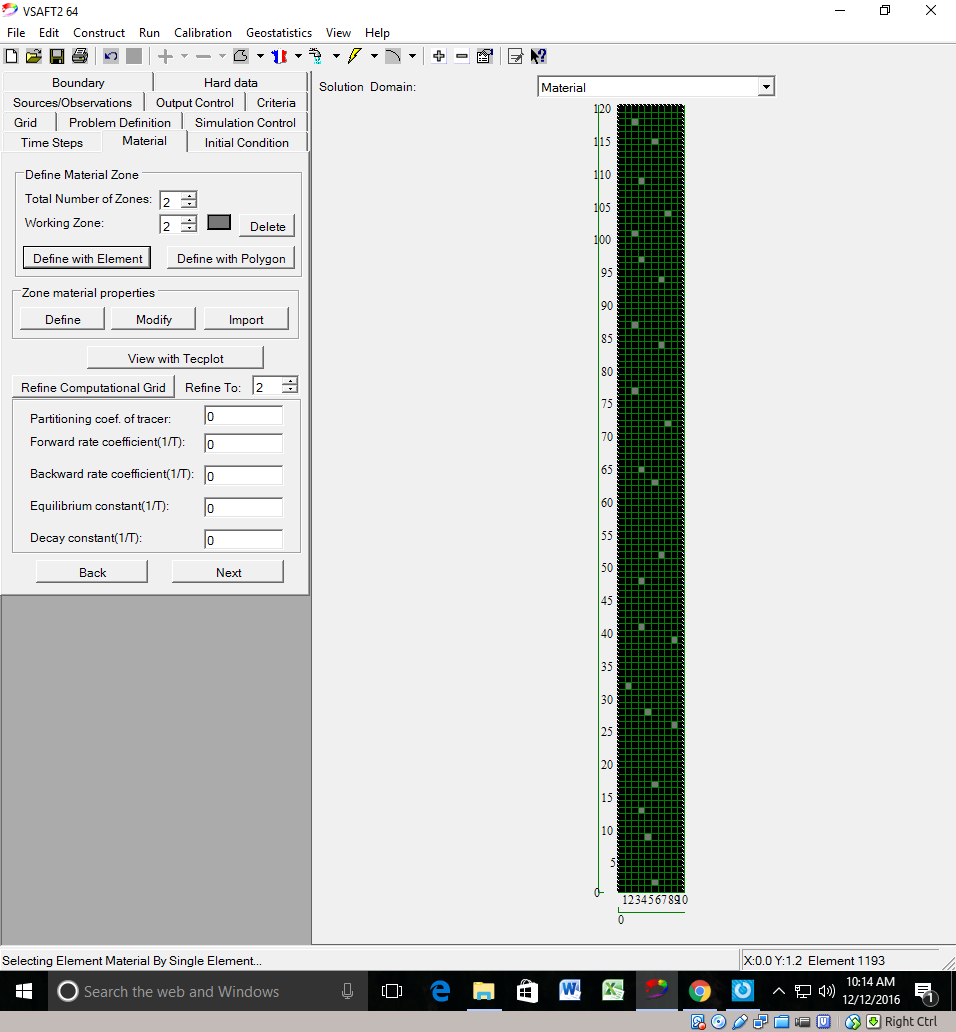
* Select **OK** after entering the properties of each material.

Define the material zones within the grid.

* All elements of the grid are by default set to zone 1. Here we will set the blocks that will be the heterogeneities to zone 2
* Set the working zone to **2** to define material 2 within the model grid



* Select **Define with Element** to change selected elements to zone 2
* Using the mouse define 24 blocks approximately randomly distributed within the model domain.

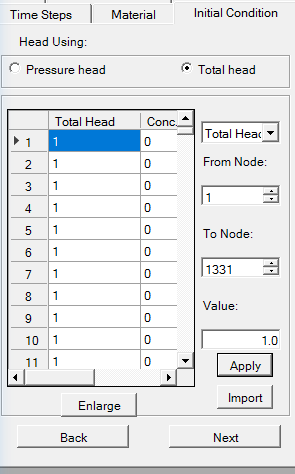


* Select **Next** to continue to the “initial condition” tab.

1. Initial conditions

Set the initial hydraulic head and solute concentrations.

* Select **Total Head**
* Enter the initial hydraulic head of **1.0** in the “value” box.
* Select **Apply.**
* The default solute concentration is 0 so we do not have to change the solute concentration.
* Select **Next** to continue to the “Source” tab.

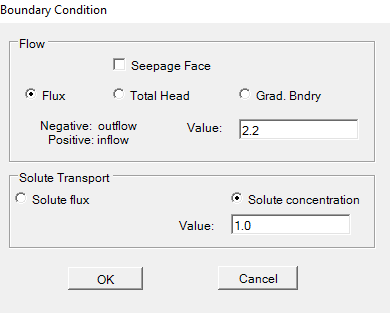


1. Boundary

Here we will set the boundary conditions.

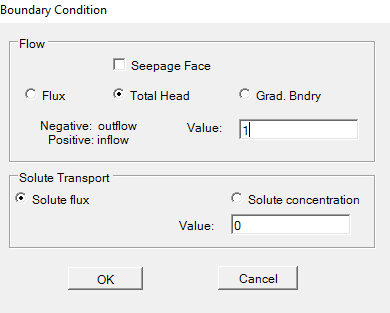
Adding constant flux boundary:

* Select **Add** from the “boundary definition” section.
* With the mouse select the beginning and the end of the top of the material domain. Also select the midpoint on the boundary so that VSAFT2 knows which direction along the boundary you wish to define. Selecting the midpoint is only required for the first boundary.
* Once the boundary is defined a pop-up window will appear.
* Select **flux** and enter the value of **2.2**.
* Select **solute concentration** and enter the value of **1.0**.
* Select **OK**



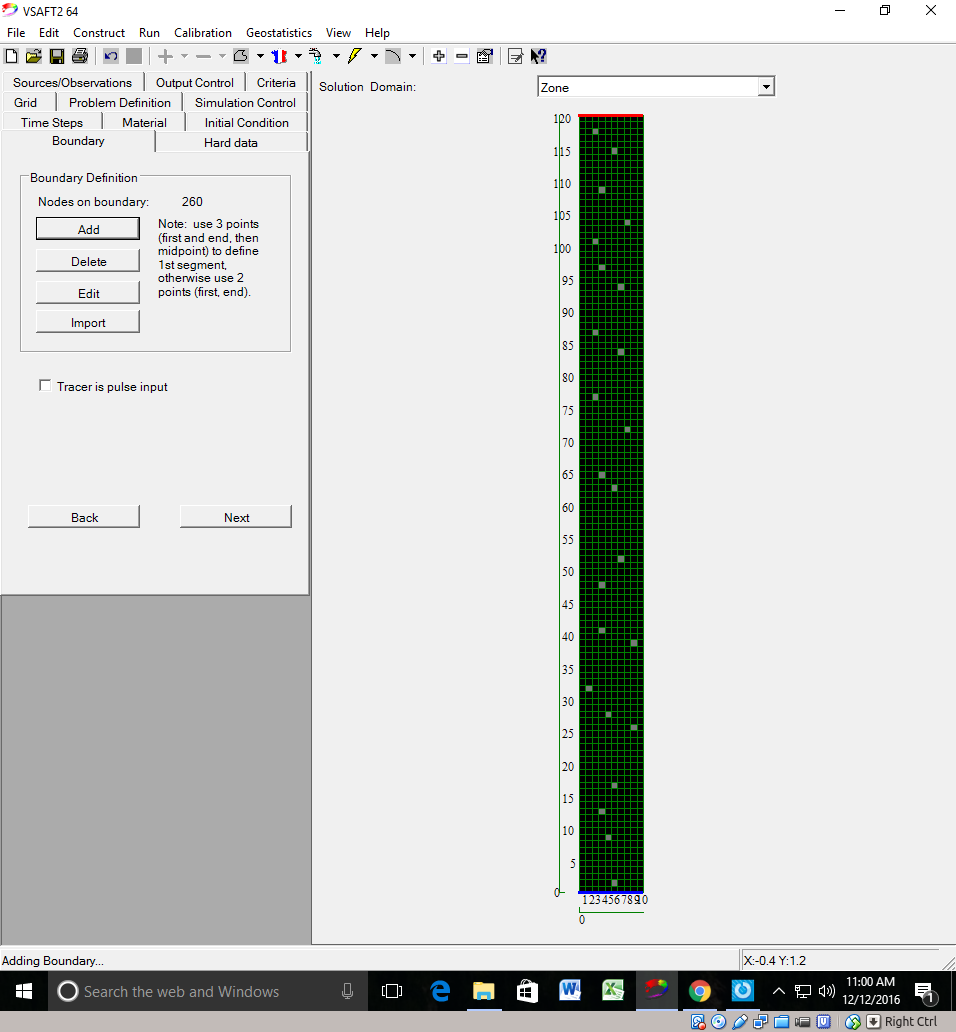
Adding Total head boundary condition:

* Select **Add** from the “boundary definition” section.
* With the mouse select the beginning and the end of the bottom boundary in the domain.
* Once the boundary is defined a pop-up window will appear to enter the boundary values
* Select **Total head** and enter the value of **1**.
* Select **solute flux** and enter the value of **0**.
* Select **OK**



The default boundary conditions are no-flow boundaries. Since these are the boundary conditions we require for the sides of the model, we do not have to explicitly define them.

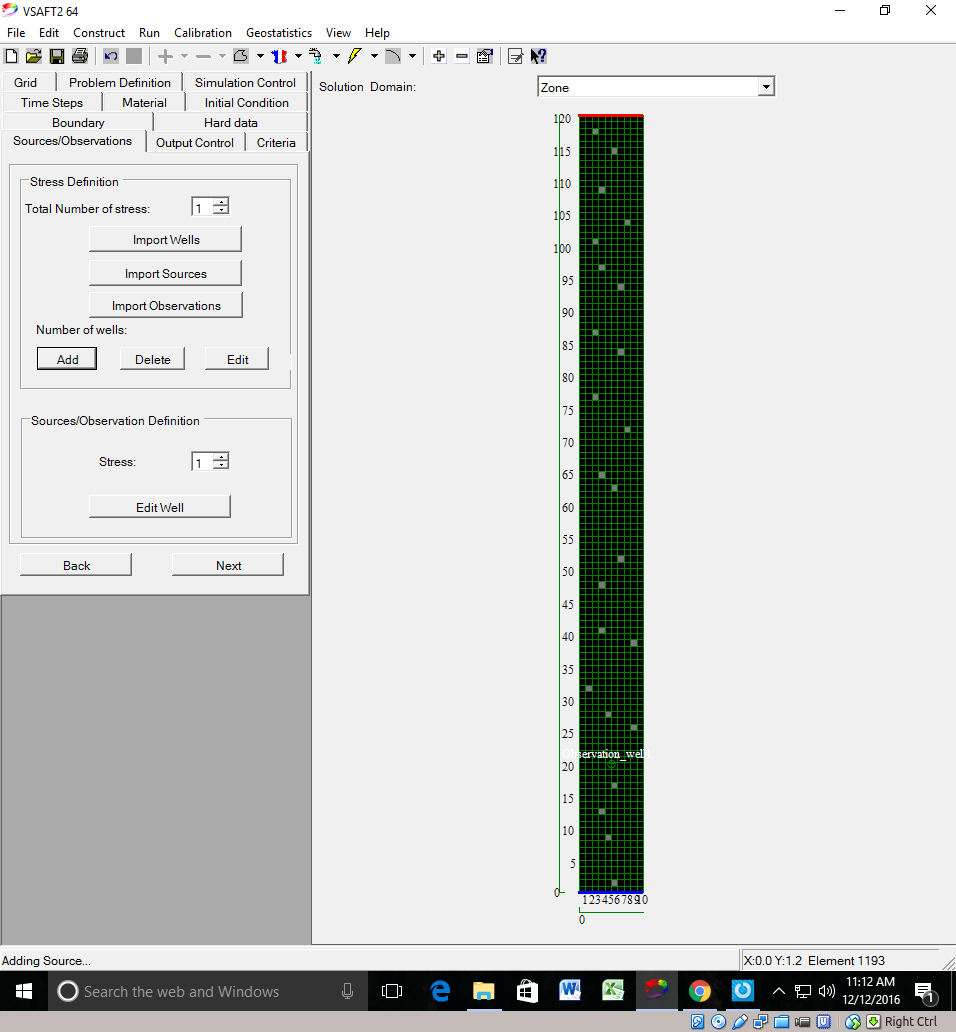
* Your model domain should know look like this.
* Select **Next** to continue to the “Sources/Observation” tab.



1. Sources/Observation

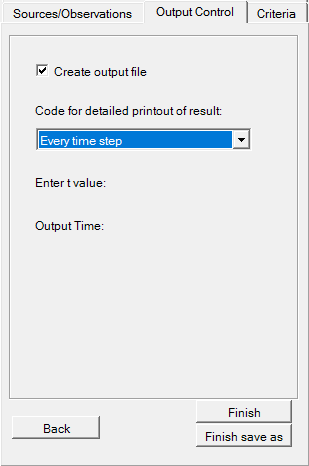
We will not add any sources however we will add an observation point.

* In the “Stress Definition” section select **ADD**. Then move the cursor over the model domain and select the center node along y= 20 cm. Name the well as **Observation\_well1** then select **OK**. Now select **Next** to move on to “Output Control”.



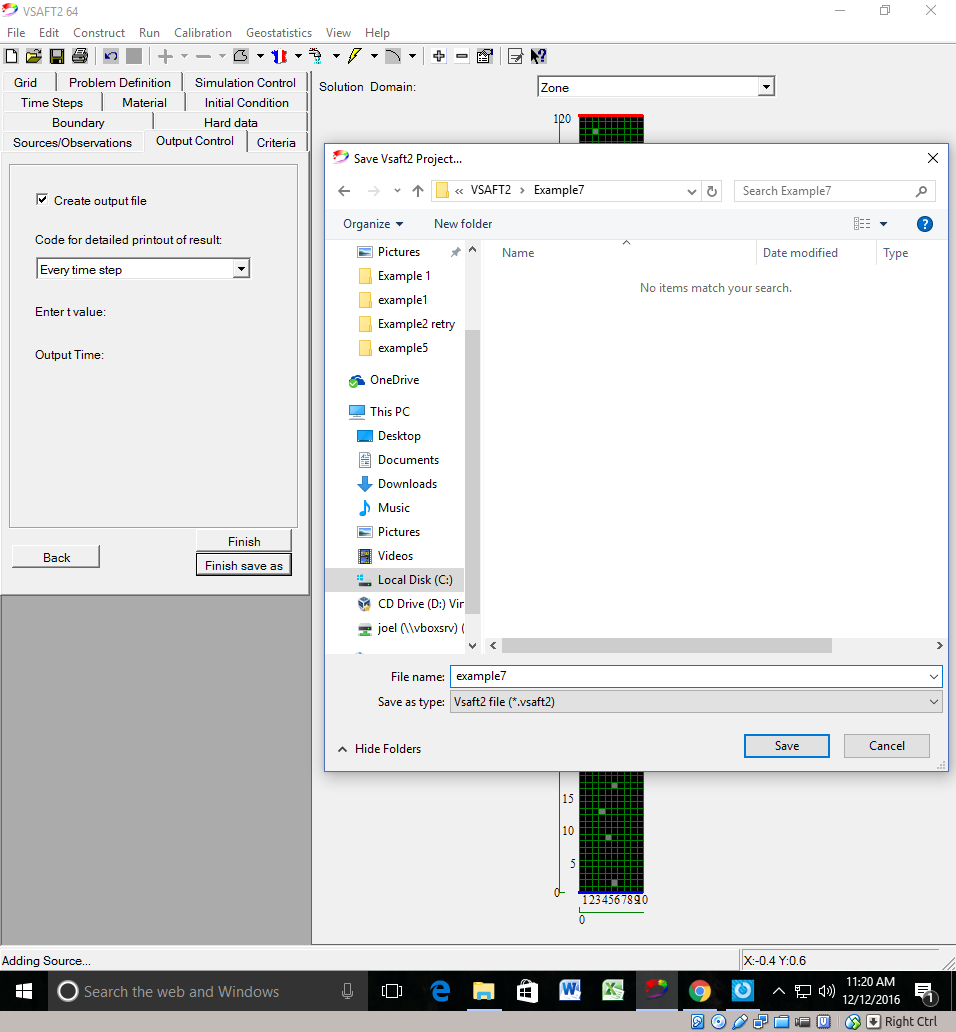
1. Output Control

* Select the checkboxto create output file to save results to a file.
* The time steps have been set to every minute from 0 to 400 minutes. Here we will select **print every time step** (default).
* Select **Finish save as** to finish the model setup and save the project to a directory.



* Save your project with the name **example7** and select **save.**

The model is now defined and the input files for VSAFT have been generated.



1. Running VASFT2

* Select **RUN** and then **VSAFT2**

1. Viewing results in TECPLOT

* To display and animate the solute concentration, follow the same steps used to display and animate the hydraulic head in example 2.

Below is a screenshot of the concentration animation.

