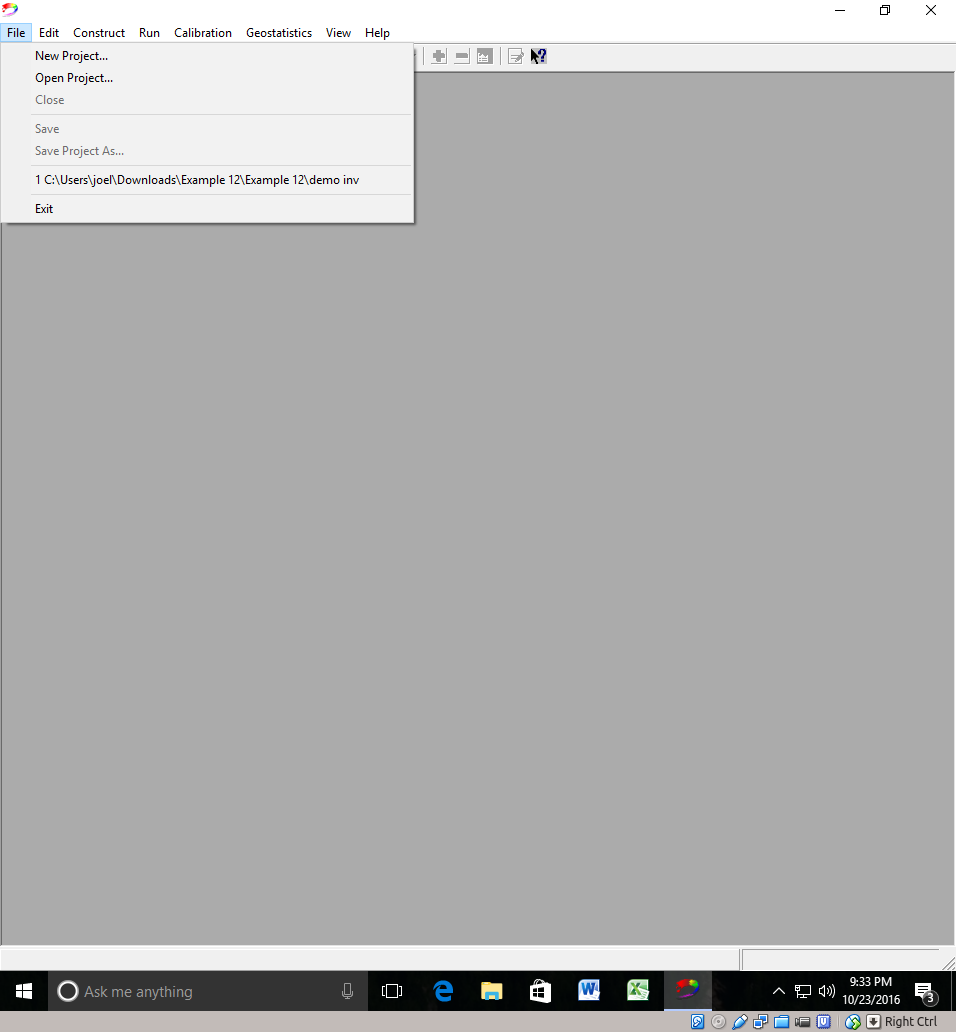
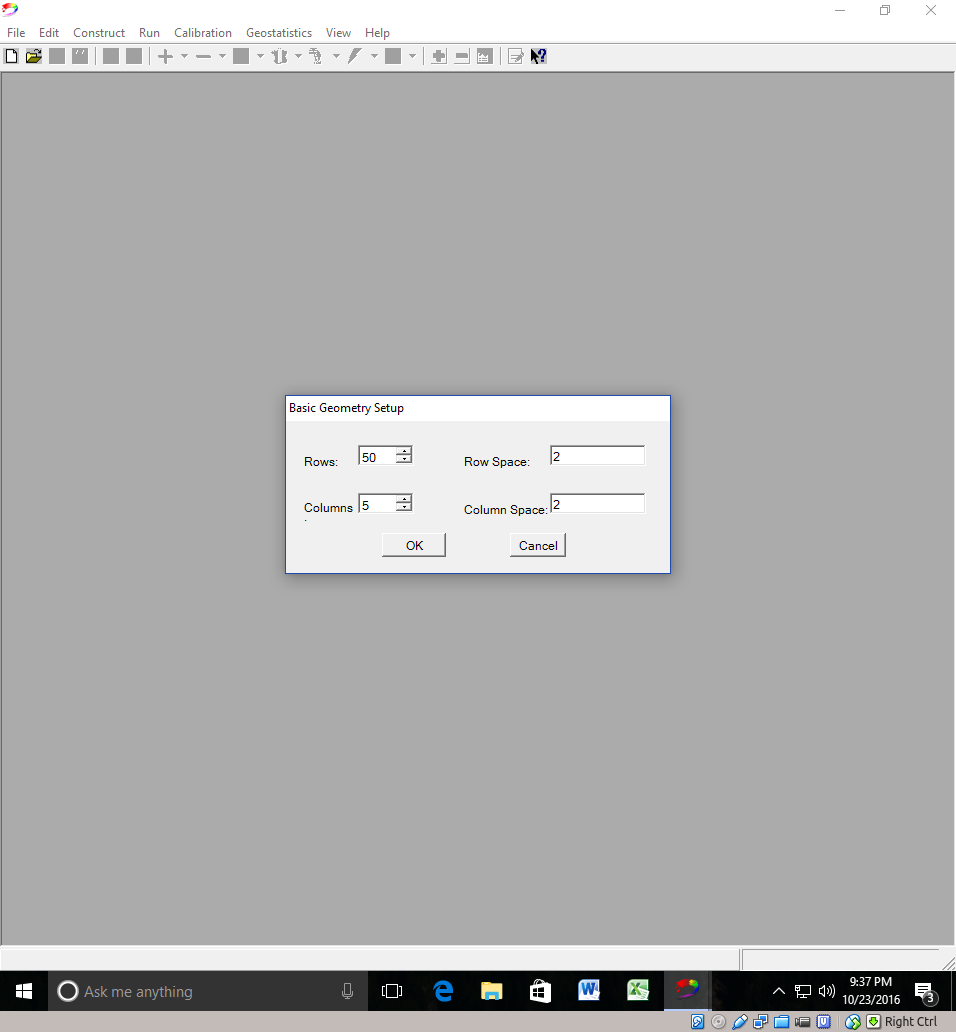
**Example 1: Vertical steady state flow in a homogeneous media.**  
The instructions for this example are provided as a list of steps with accompanying screen captures. These examples where made using VSAFT2 version 2015.1.6184\_64 with a GUI version 2016.10.02 and all graphs where created using Tecplot 360 2010 Build 12.2.9.9077.

1. New Project: Select “New Project” under the file menu.



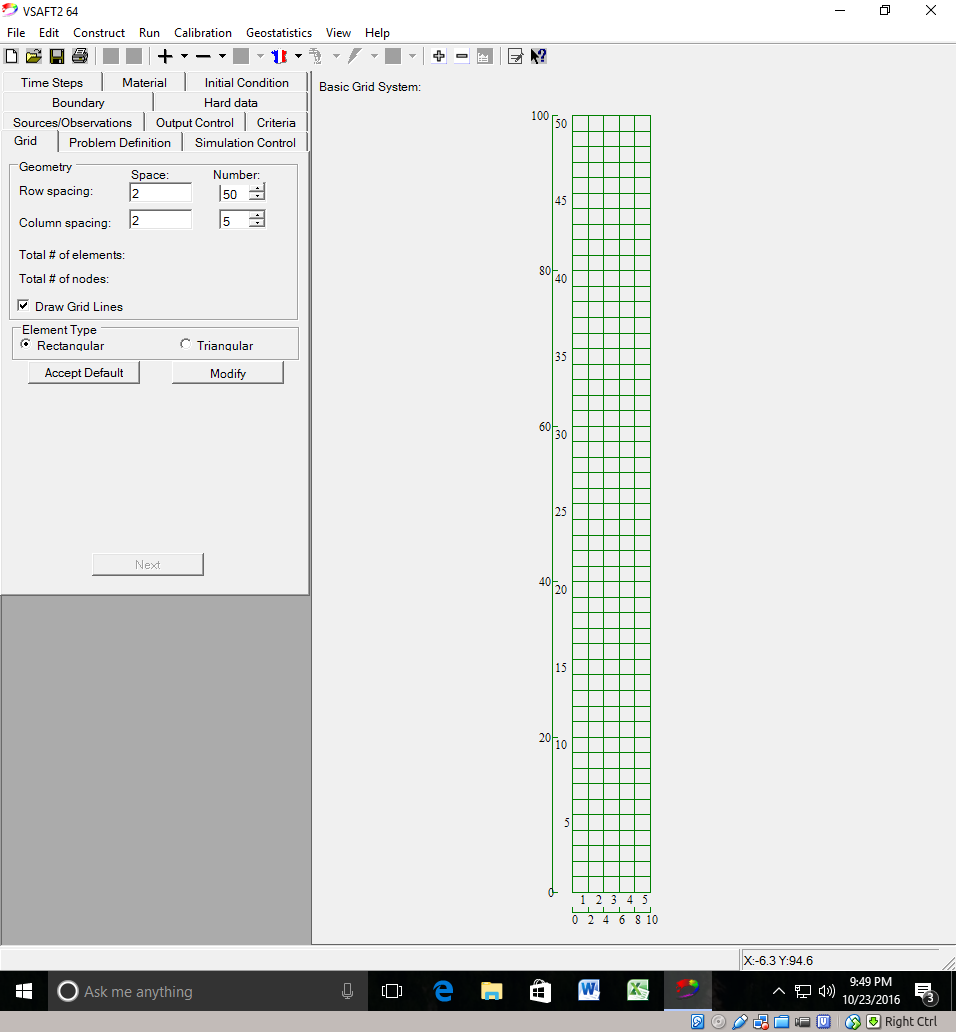
1. Problem Definition:

* Define the initial grid dimensions: Set the number of rows to **50**, columns to **5**, and change the row and column spacing to **2**.
* Then select **OK**.



1. Edit Grid:

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

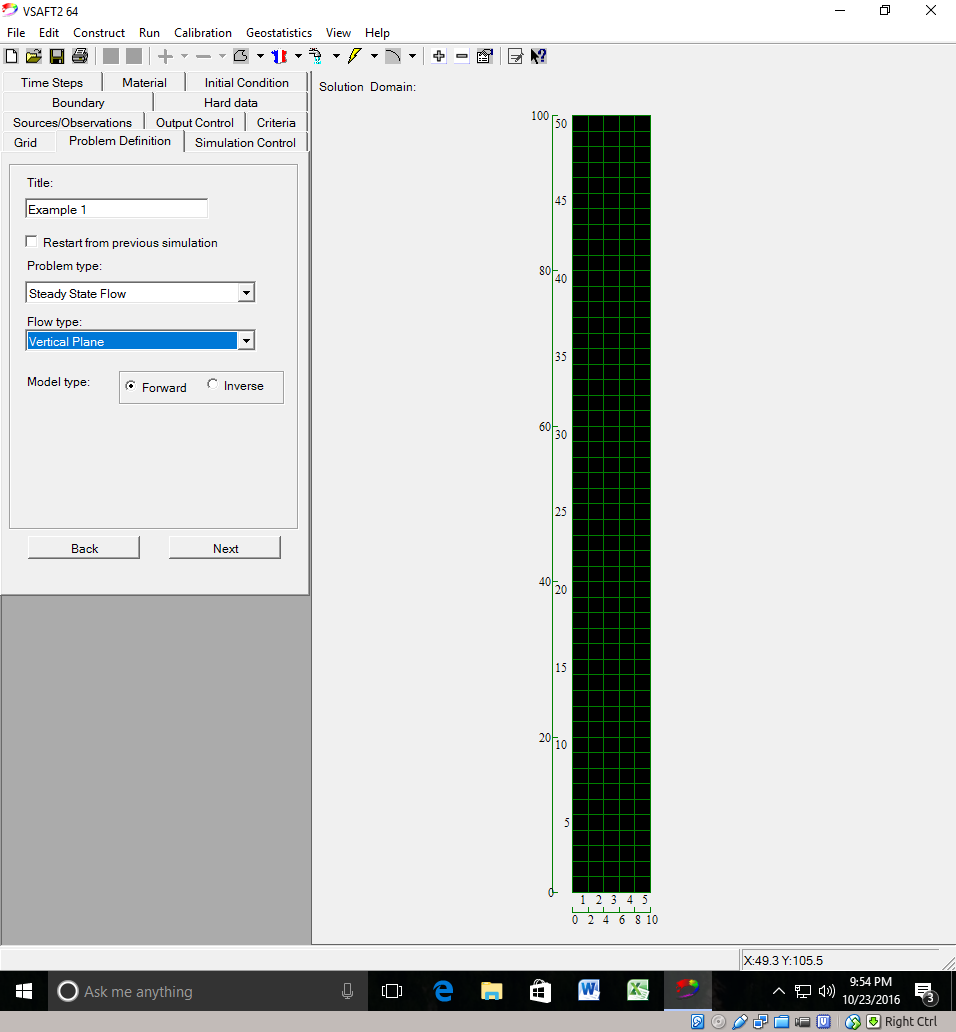
  
  
For this example we will accept the grid without editing the row or column spacing or adding additional rows or columns.

* Click **Accept Default.**

1. Name and Define Project

Enter a title in the “TITLE” box. This is for record keeping purposes and to assist in remembering the details of the model.

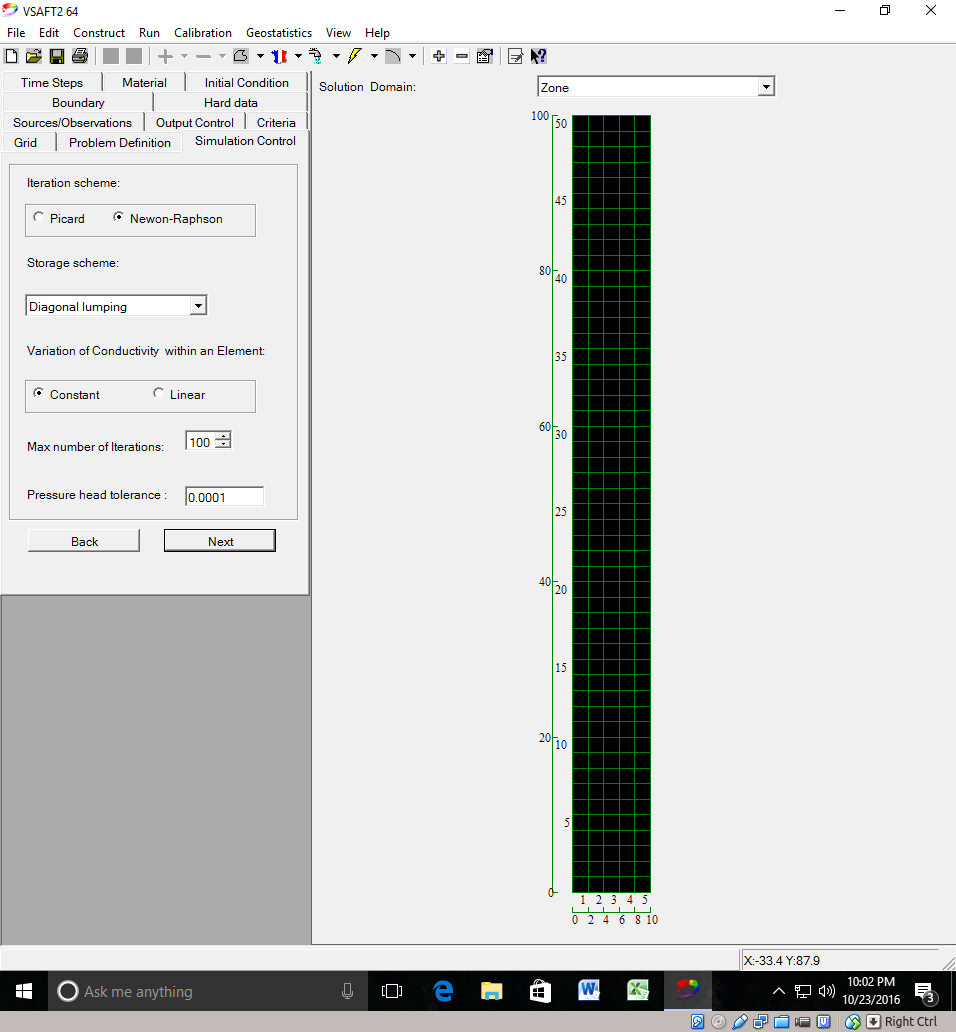
* Use a descriptive title. Use the Problem type drop down menu to select **Steady State Flow**.
* Use the Flow type drop-down menu to select **Vertical Plane** then check the **forward model** box.
* Select **Next** to continue to the “simulation control” tab.



1. Simulation Control

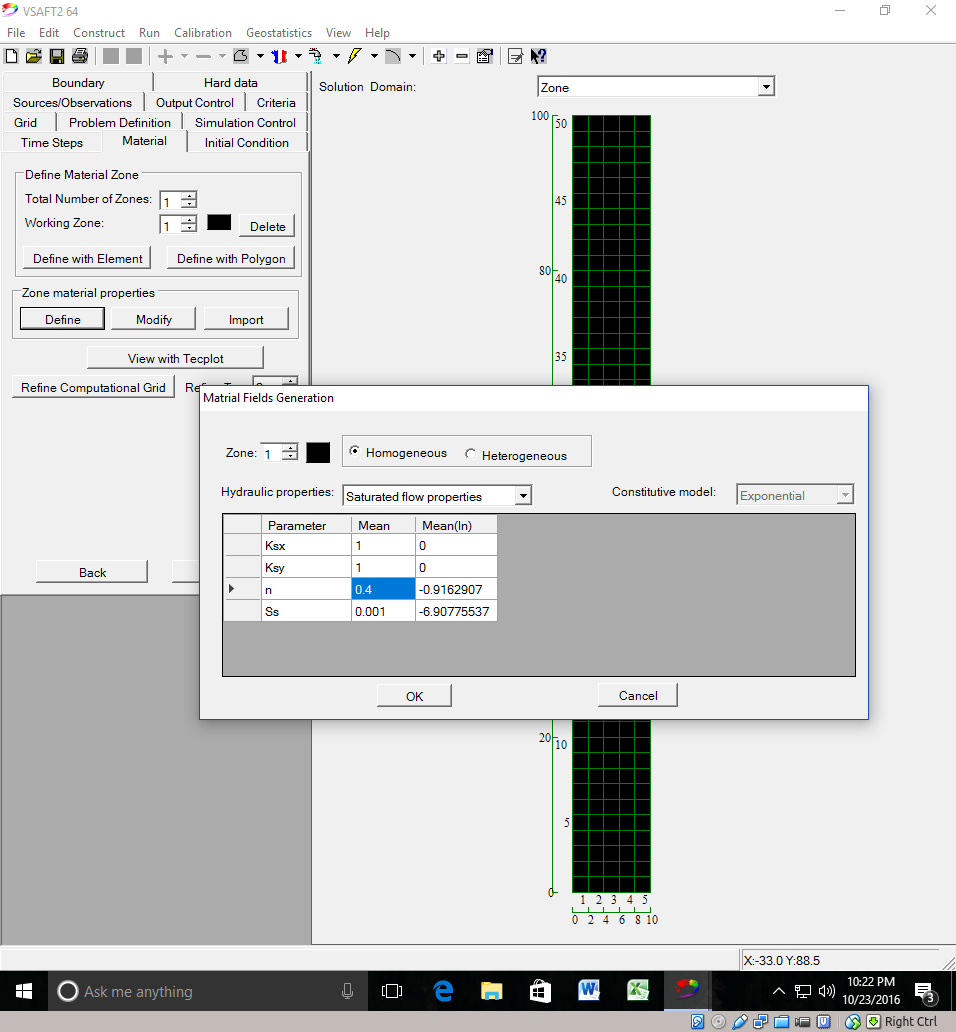
We will accept the default simulation control parameters for this example.

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



1. Materials

* Under the materials tab and in the Zone material properties select **Define**.

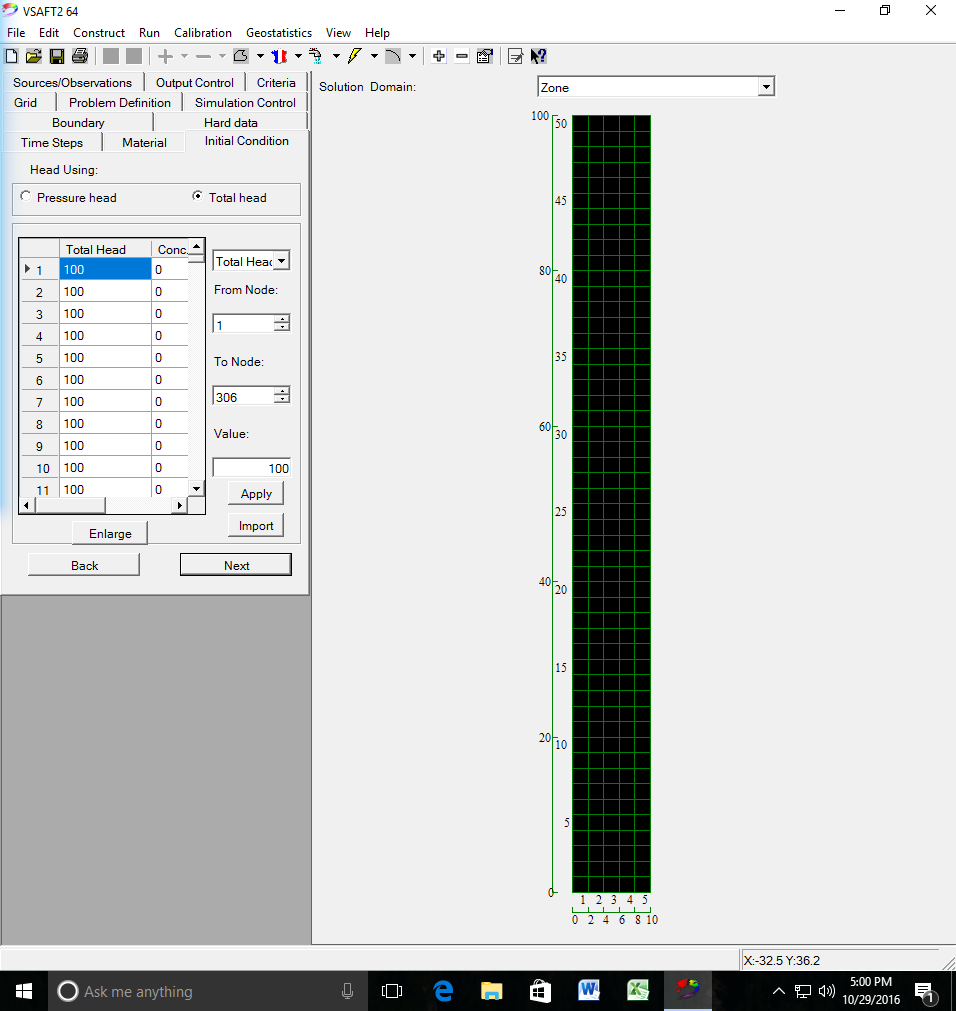


* Enter the material properties: Select a **Homogeneous** for a homogeneous porous matrix for zone 1.
* Enter the values for the saturated flow properties under the “Mean” column. In this example set an isotropic hydraulic conductivity value of **1**, set porosity (n) to **0.4** and the specific storage (Ss) to **0.001**.
* Then select **OK** and select **Next** to continue to the Initial condition tab

1. Initial Conditions

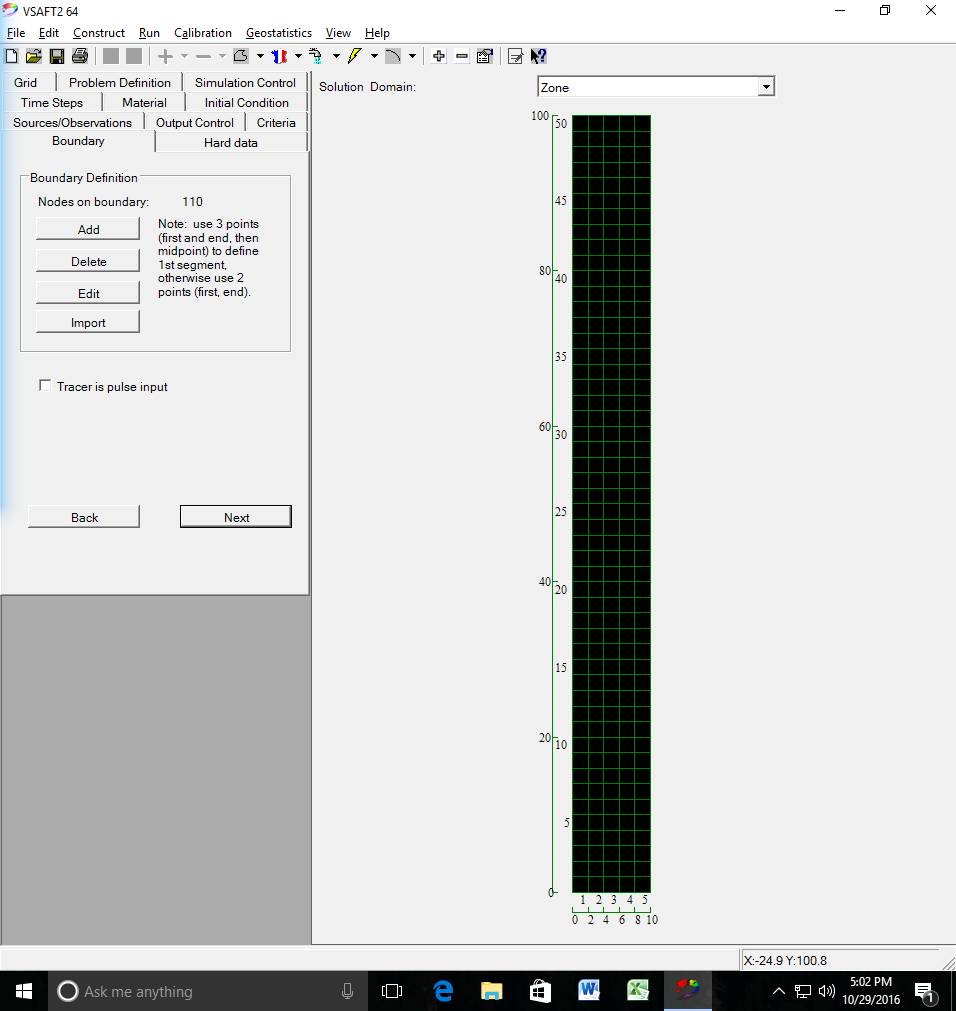
Set the initial hydraulic head and solute concentrations.

* Select **Total Head** then enter the initial hydraulic head of 100 in the value box.
* Next Select Apply then select **Next** to continue to the “boundary” tab.

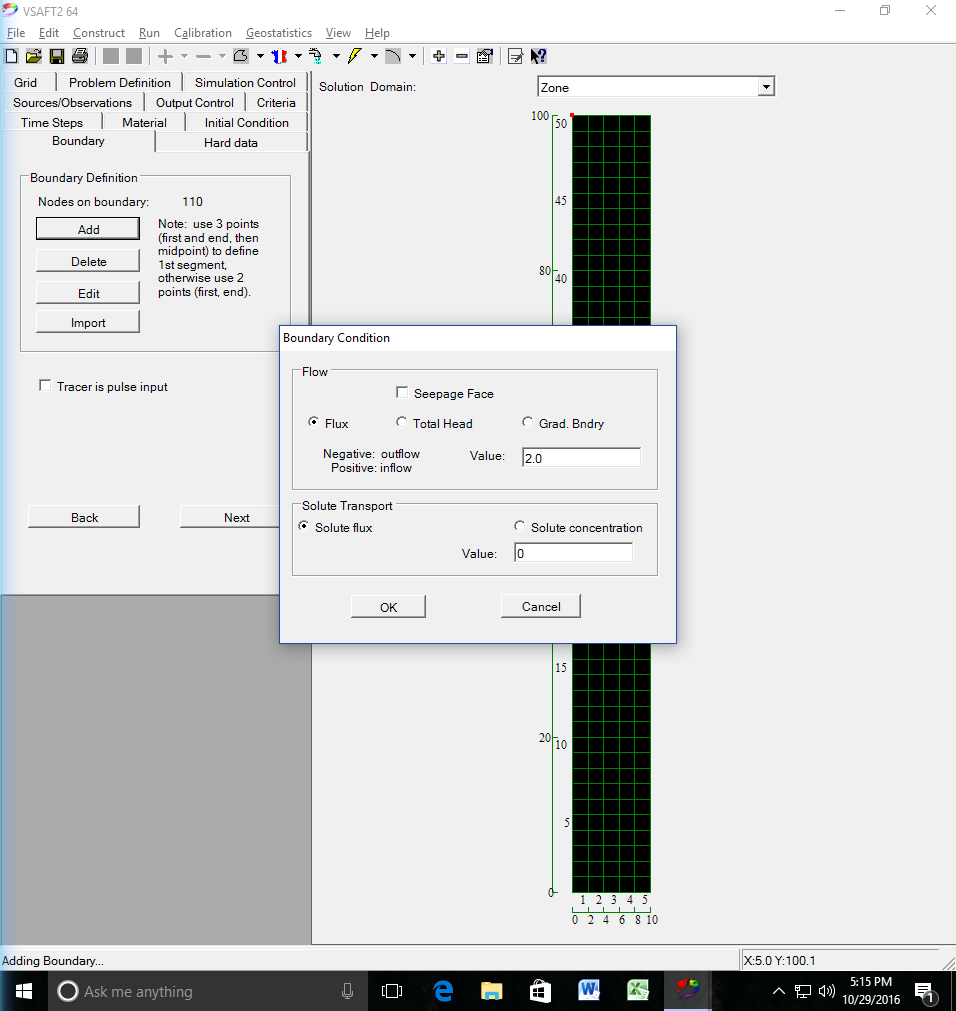


8. Boundary

* To set the boundary conditions select **Add** from the “boundary definition” section and define the top prescribed head, constant concentration boundary.



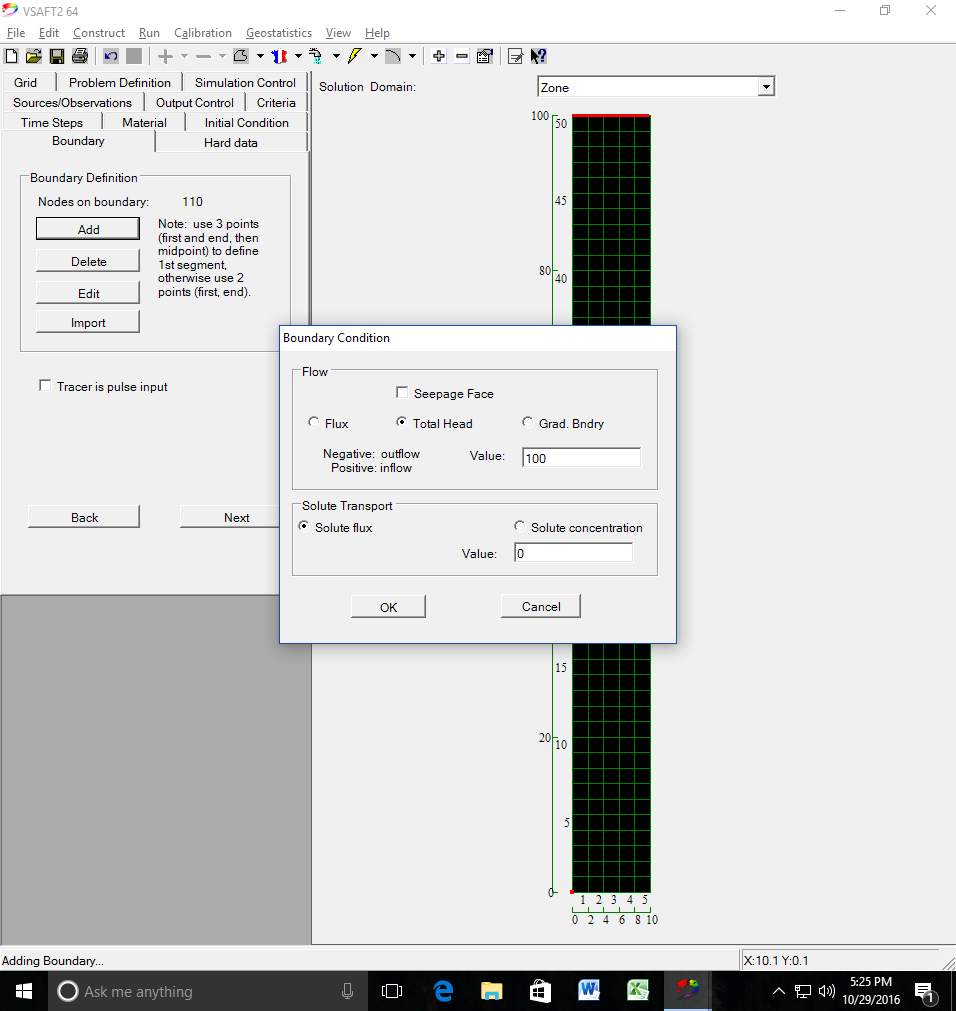
* With the mouse select the beginning and the end of the first boundary (at the top).
* Also select the midpoint on the boundary so that VSAFT2 knows which direction along the boundary you wish to define. This step is only required for the first boundary.



* Once the boundary is defined a pop-up window will appear to enter the boundary values.  
  Under the “Flow” section, select **Flux** and enter the value of **2.0**.
* Then select **OK**. This boundary condition will be displayed as a red line.

Next, select **Add** again from the boundary condition window and define the bottom of the column a prescribed head and constant concentration boundary.

* With the mouse select the bottom boundary of the column. In the Boundary Condition popup window set the boundary condition type to **Total Head** and enter a value of **100** then select **OK**.
* Once you select OK the boundary condition will appear as a solid blue line



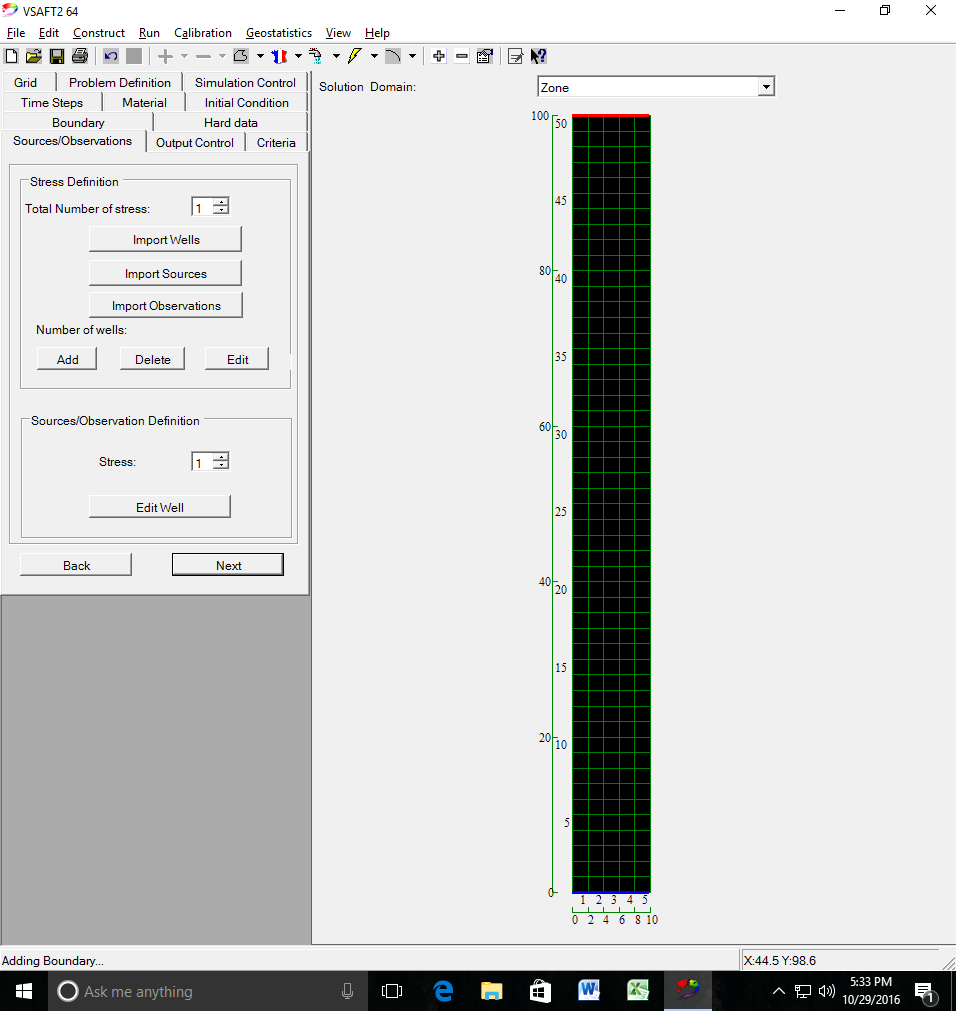
The default boundary conditions are zero which represent no flow boundaries therefore because we want to define the two vertical edges as no-flow boundaries we do not need to explicitly define them.

* Click **Next** to continue to the “Sources” tab.

9. Sources

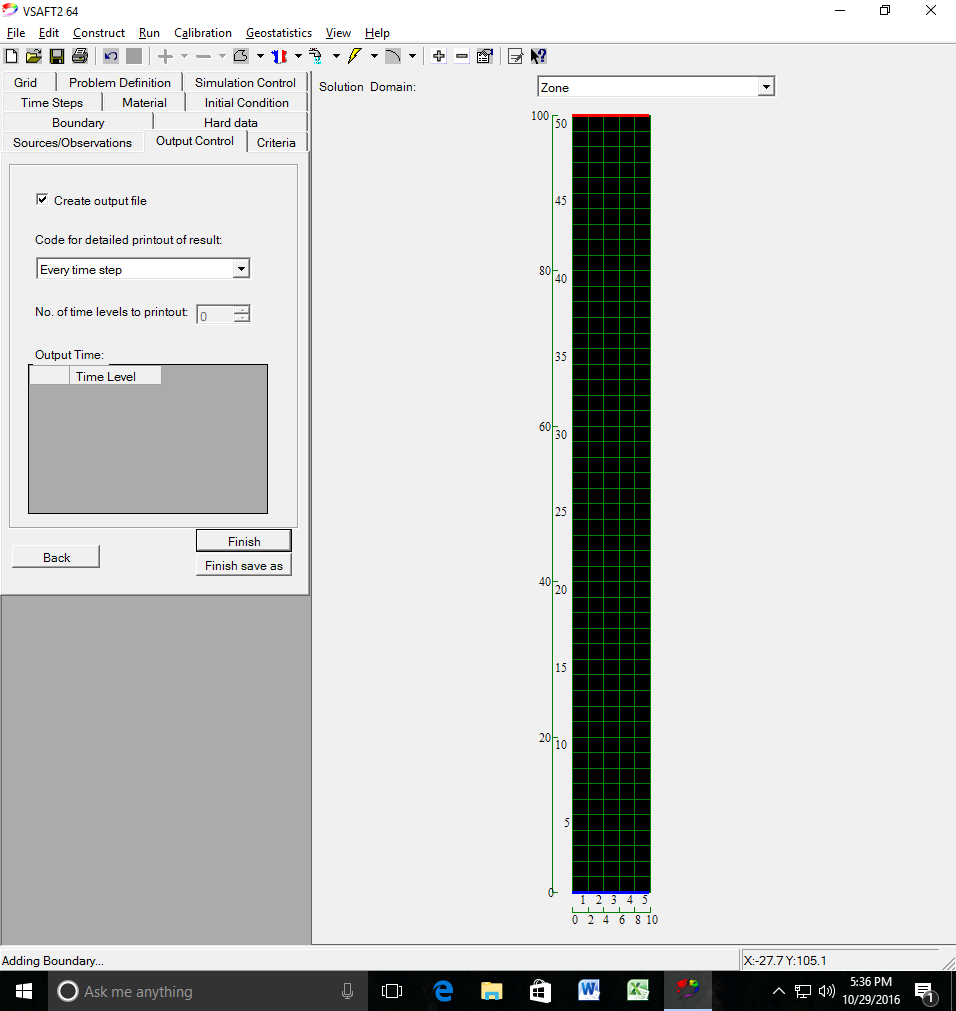
We will not be adding any sources or sinks for this case. We also will not be adding any observation wells.

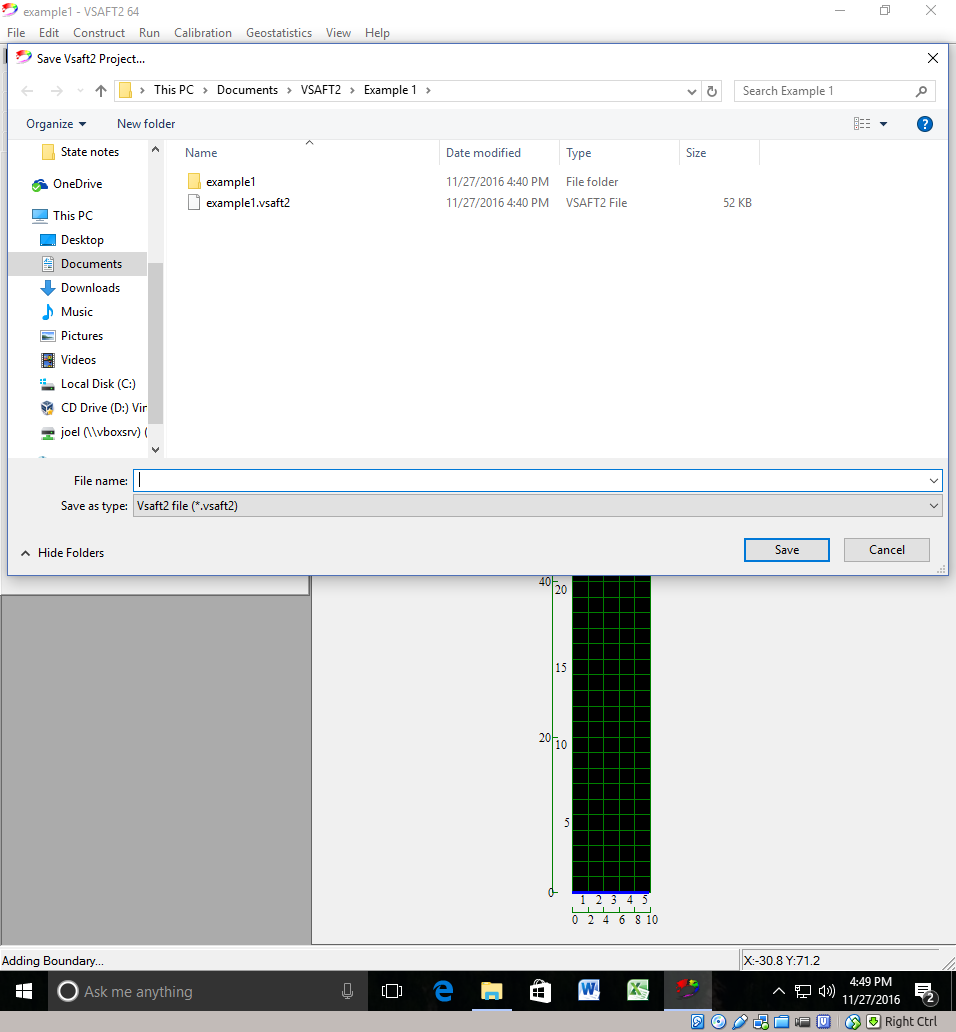
* Therefor select **Next** to move to “Output Control” tab.



10. Output Control

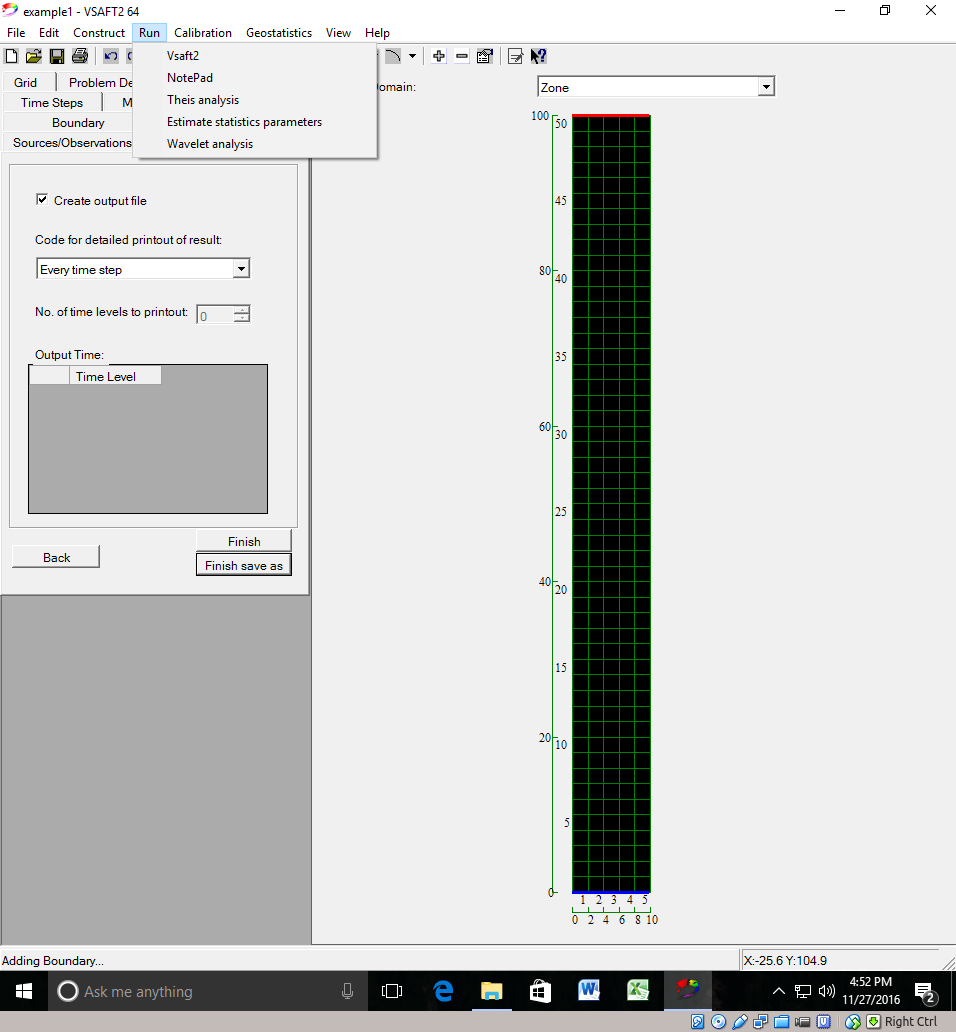
* First make sure there is a checkmark in the Create output file check box. This file will allow us to save our results to a file.
* Select **Finish save as** to finish the model setup and save the project to a directory.





11. Running VASFT2

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



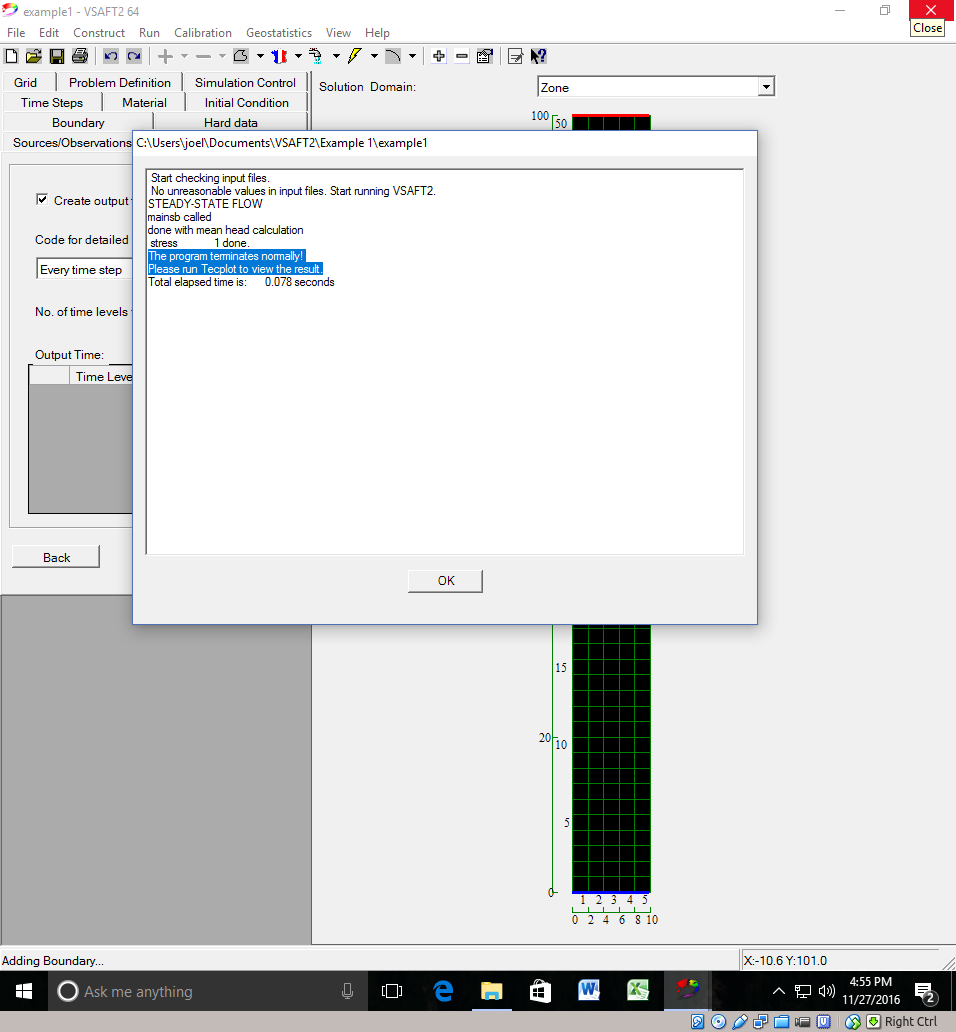
When the program has finished it will display the dialog box seen below.

The window will say:

**“**The program terminates normally!

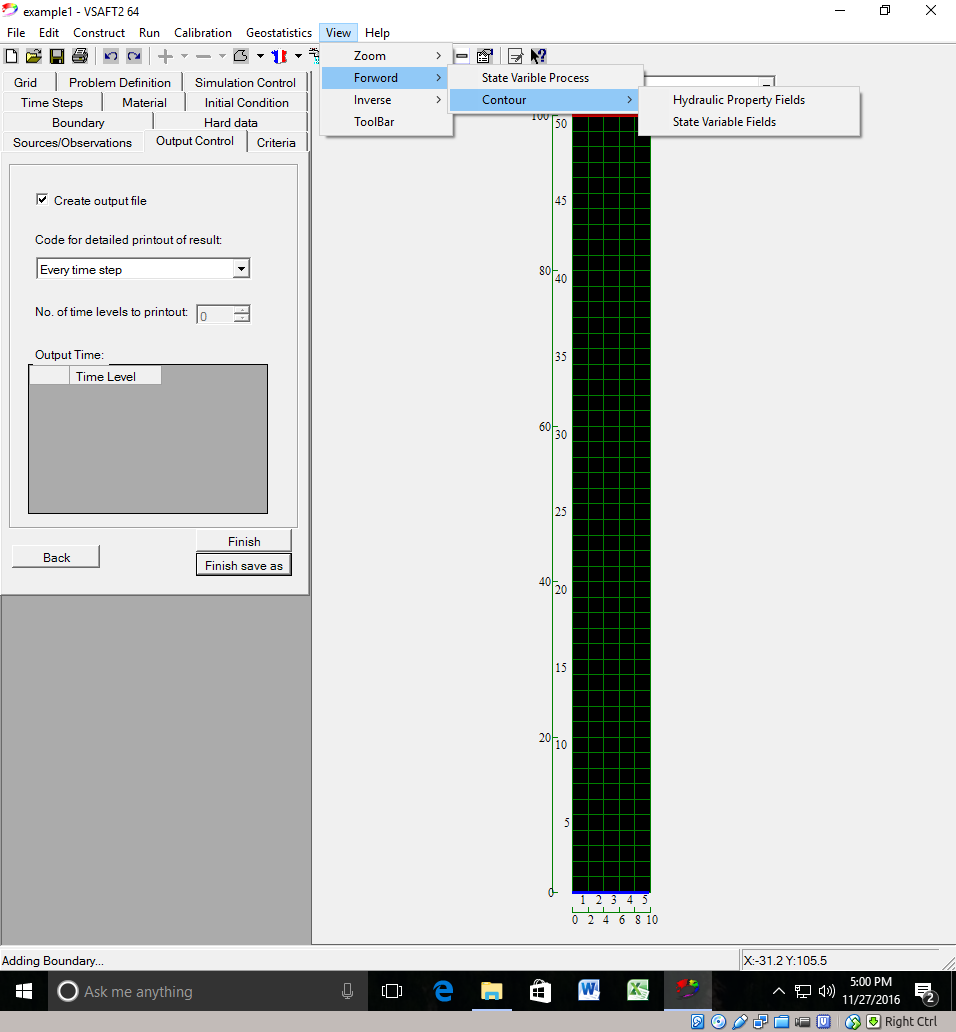
Please run Tecplot to view the result.**”**

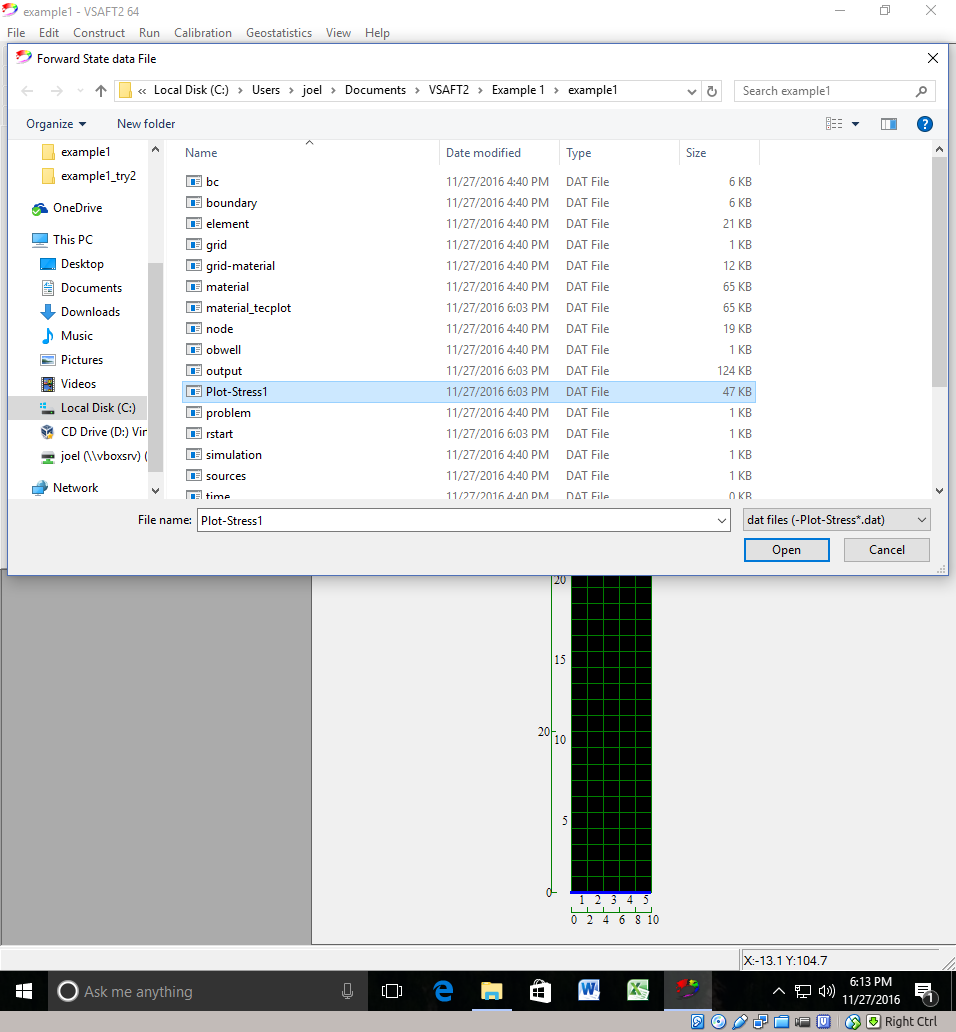
* Click **OK** to continue.

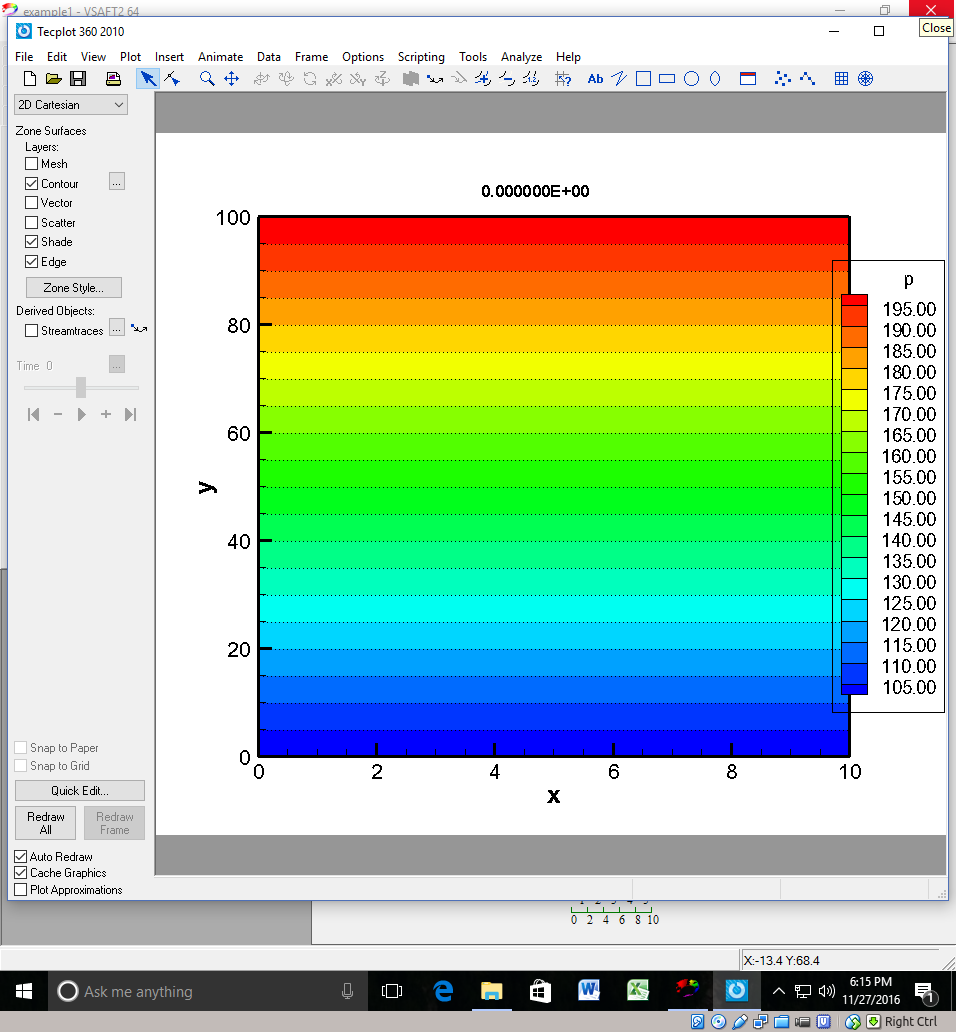


12. Viewing results in TECPLOT:

* To view results select the **View** menu in the menu bar.
* To view the hydraulic head distribution selects **View-> Forward->Contour ->State Variable Fields**.
* A list dialog box of the available outputs will be displayed. Select **Plot-Stress1** which contains the hydraulic head data. This will automatically start TECPLOT and display the hydraulic head as a multi colored contour diagram.







13. Tecplot 360 results

The axis can be adjusted by selecting **Plot** menu and then **Axis…** in TECPLOT. After some adjustment your graph should like the screenshot below. You can also change the variable being plotted by selecting **…** next to the “Contour” checkmark box.

