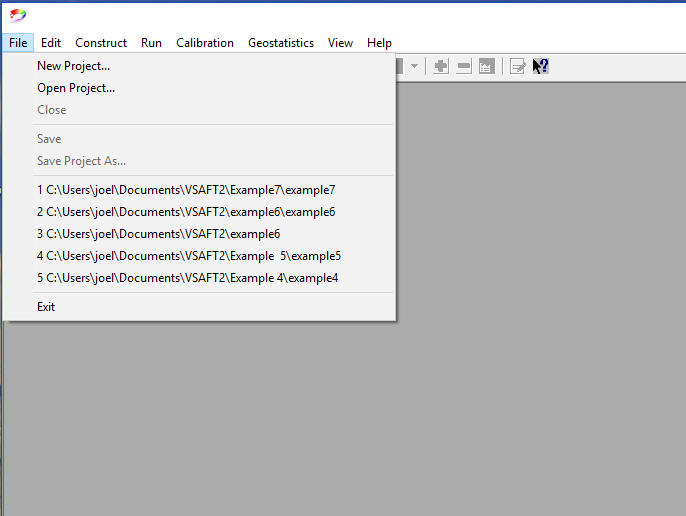
**Example 8: Well-Posed one dimensional Horizontal Flow Problem**

This example show how to set up and solve a well posed 1-D inverse problem. The problem is divided into two parts: first the forward model is used to generated observations at specific well sites. These observations will then be used in an inverse model to predict the domain conditions of the forward model. The instructions for this example are provided as a list of steps with accompanying screen captures.

**Part 1: Generating observations well data using:**

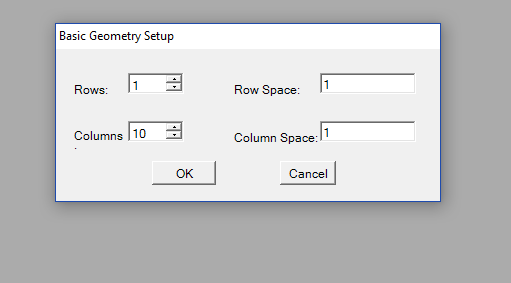
1. New Project

* Start a new project by selecting **File** then **New Project**.

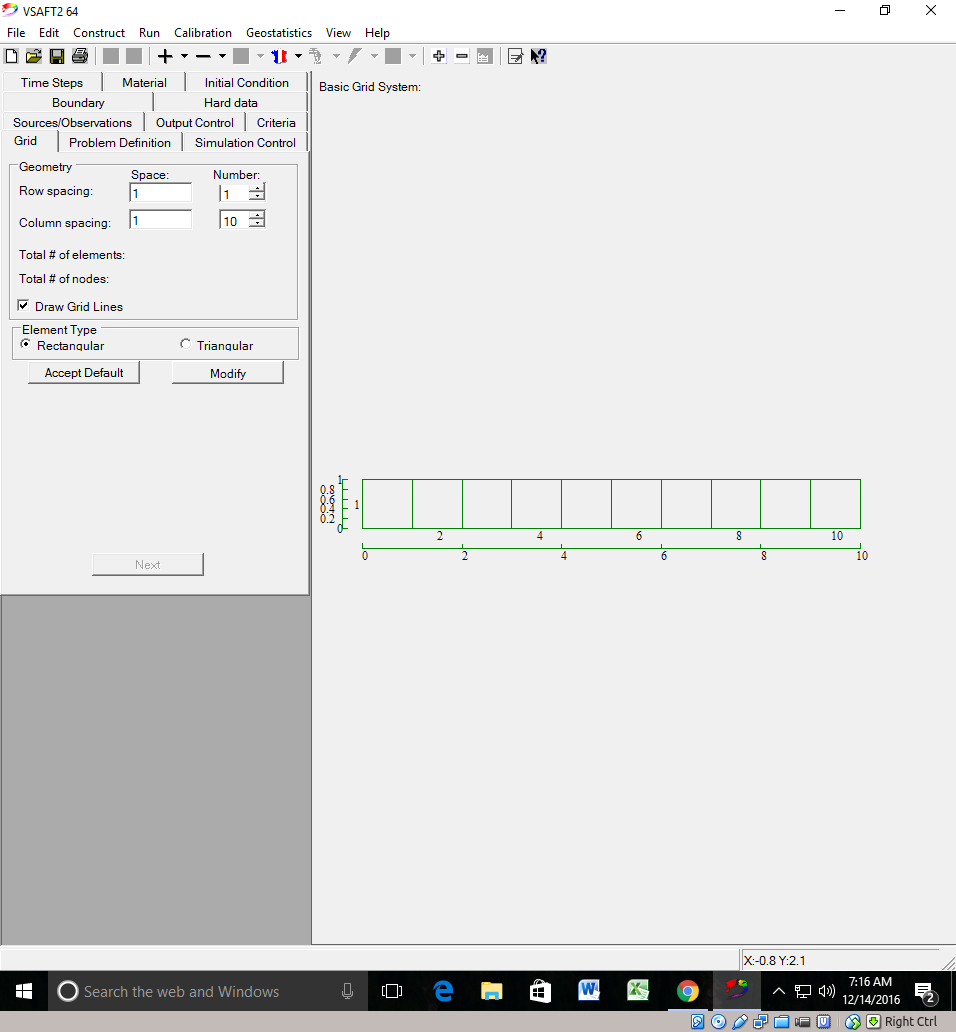


Define the initial grid dimensions (you can edit this later).

* Change the value for the number of rows to **1**
* Change the value for the number of columns to **10**.
* Change the row and column spacing to **1**.
* Select **OK**



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

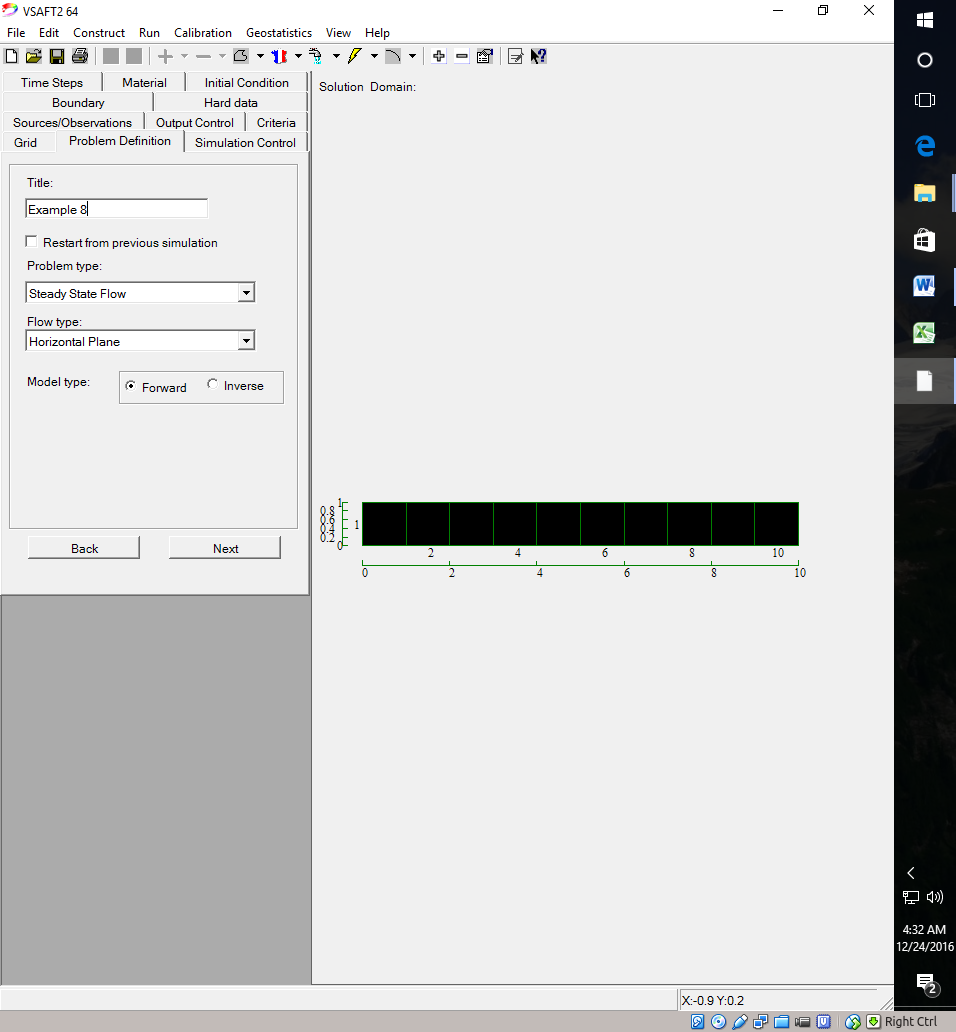


For this example we will accept the grid without editing the row or column spacing or adding additional rows or columns.

* Select **Accept Default**
* Select **Next** to advance to the “problem definition” tab.

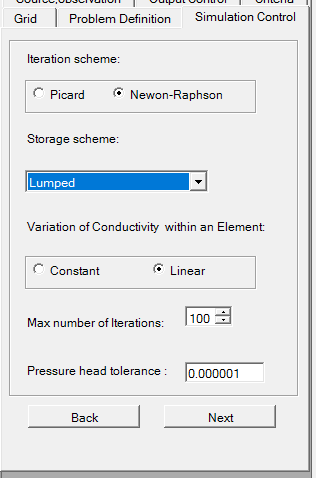
1. Problem Definition

* 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 **Horizontal Plane**
* Choose model type as **Forward**
* Select **Next** to continue to the simulation control tab.



1. Simulation Control

* Select the **Lumped** storage scheme.
* Select the **Linear** under the “variation of conductivity within an Element” section.
* Set the **Pressure head tolerance** to **0.000001**
* Select **Next** to continue to the Material tab.



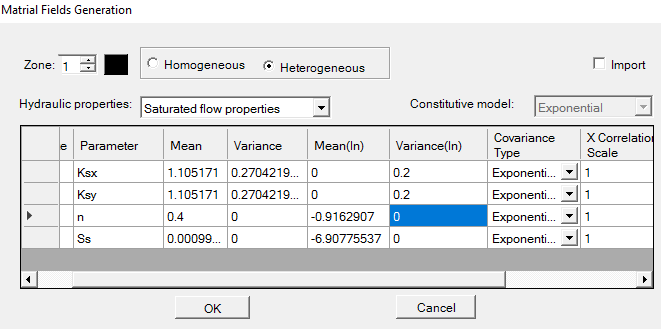
1. Materials

This example uses random material.

* Click **Define** in the “Zone Material Properties” section
* Choose **Heterogenous** for representing random material.
* In the Hydraulic properties box select **Saturated Flow properties**.

Enter the X and Y correlation scale and the variance for Ksx, Ksy.

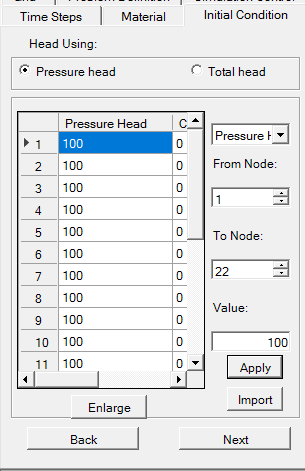
* X correlation scale = **1**
* Y correlation scale = **1**
* Variance (ln) = **0.2**
* Mean (ln) = 0
* Select **OK**
* Select **Next** to continue to the “Initial condition” tab



1. Initial conditions

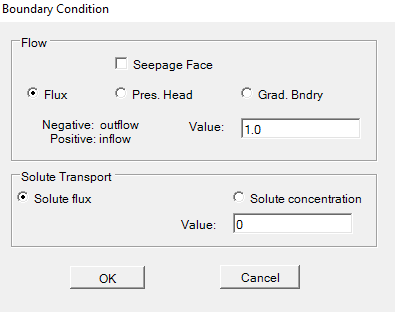
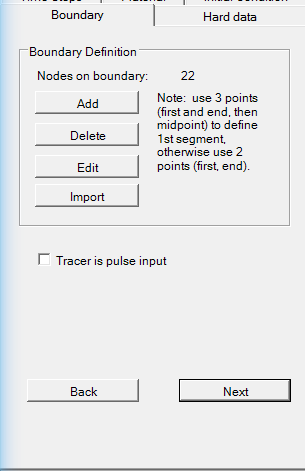
Set the initial hydraulic head

* Select **Pressure Head**
* Enter the initial hydraulic head of **100.0** in the “value” box.
* Select **Apply.**
* Select **Next** to continue to the “initial condition” tab.



1. Boundary

Here we will set the boundary conditions: the left side as a prescribed flux and the right side as a prescribed pressure.

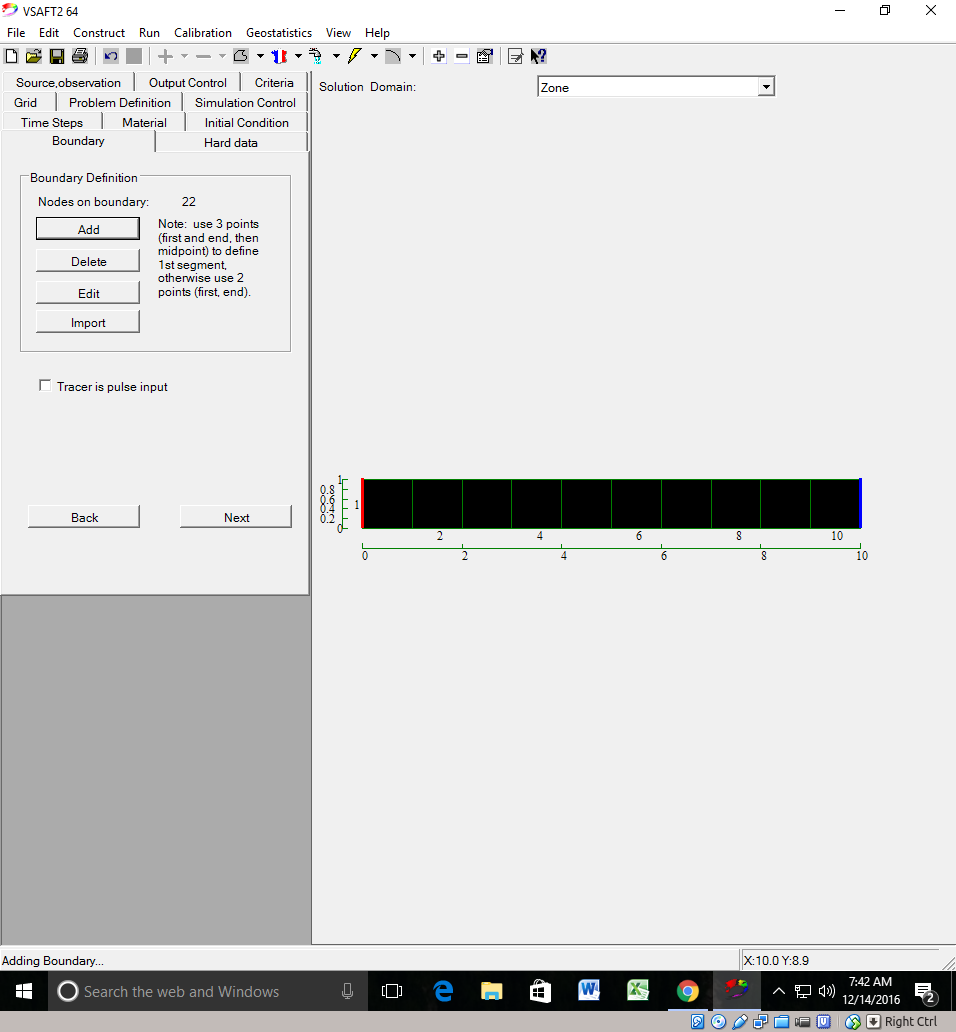
* Select **Add** from the “boundary definition window”.
* With the mouse select the **beginning** and the **end** of the boundary on the left of the column.
* Also select the midpoint on the boundary so that VSAFT2 knows which direction along the boundary you wish to define. This step is only required for the first boundary.

Once the boundary is defined a pop-up window will appear to enter the boundary values.

* Select **flux** and enter the value of **1.0**.
* Select **OK**

Add another boundary condition to the right side of the domain.

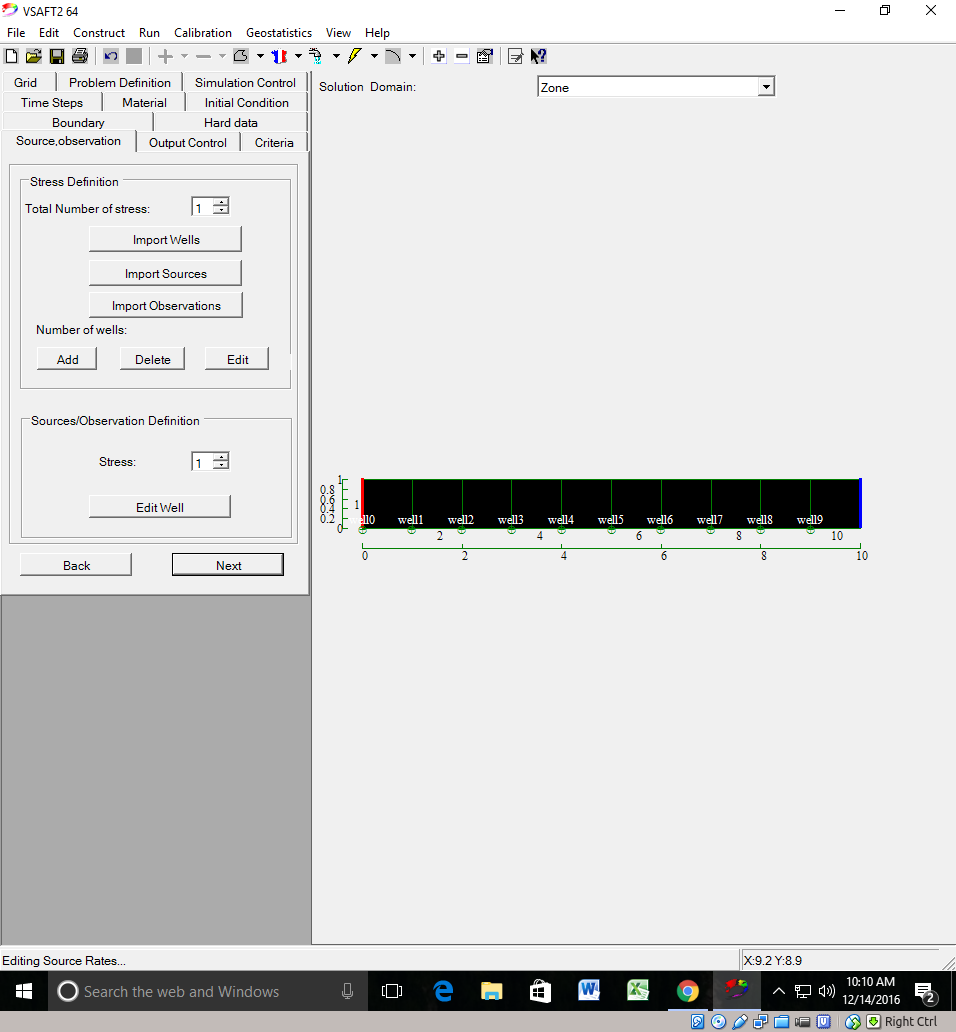
* Select **Add** from the “boundary definition window” and define the right prescribed head boundary.
* With the mouse select the beginning and the end of the boundary on the right.
* Once the boundary is defined a pop-up window will appear to enter the boundary values
* Select **Pres. Head** and enter the value of **100**.
* Select **OK**
* Select **Next** to continue to the “Source/observation” tab.



1. Source/observation

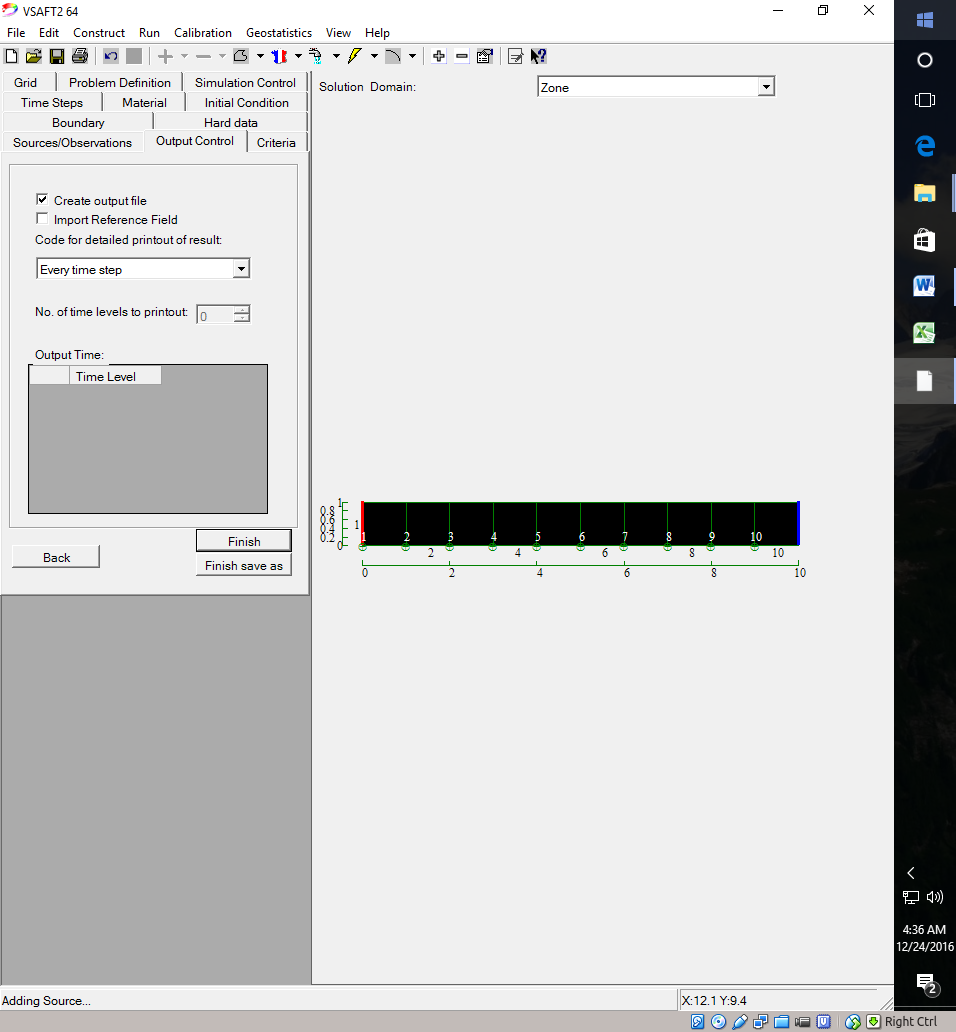
We will not need to add any sources because we are describing natural flow condition.

Assign observation wells at each node on the bottom lines except the constant head boundary. These observation wells are shown with circles.

* Select **Add** under the “stress definition” section.
* Click the bottom part of the domain and create and add 10 observation wells totally.
* Name each well.
* Select **Next**.

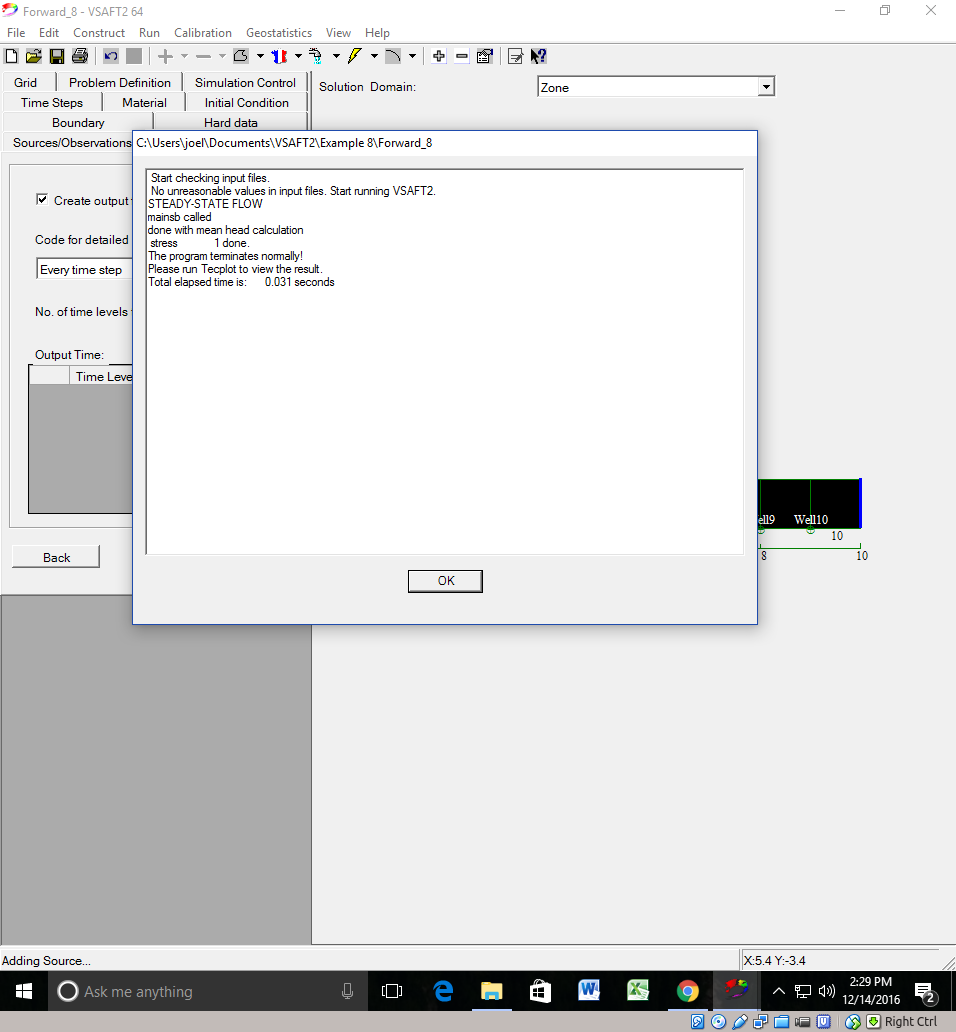
1. Output Control

* Make sure the **Create output file** box is checked.
* Leave the other settings as defaults.
* Choose **finish save as** to save the project.
* Save the project as **Forward\_8.**



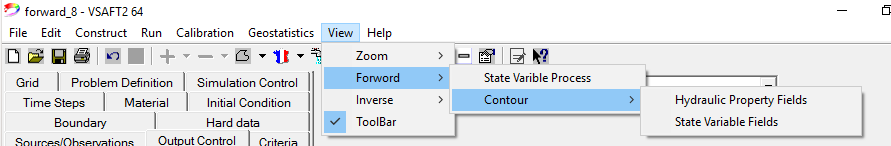
1. Running VASFT2

* Select **RUN** and then **VSAFT2**
* When finished, the screen will look like this.

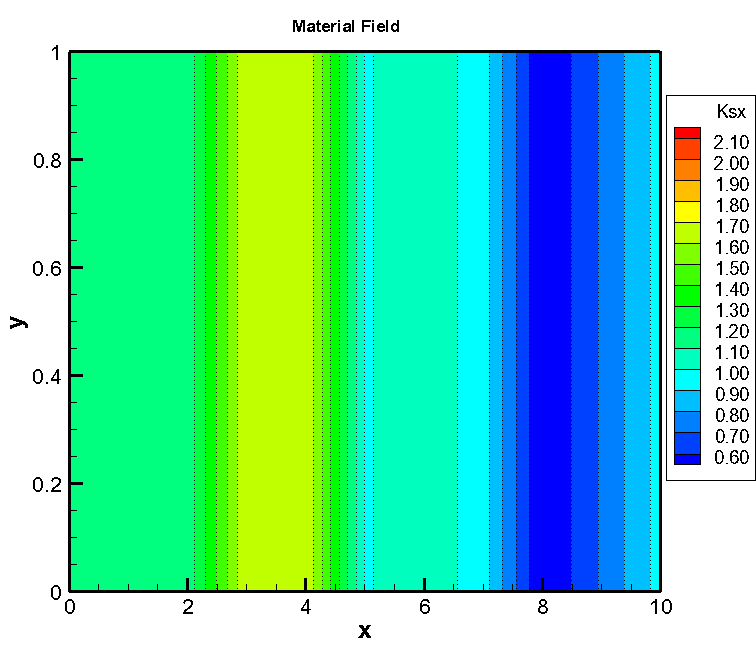


1. Plotting the hydraulic Conductivity

* They hydraulic conductivity can be plotted by selecting **View->Contour->Hydraulic Property Fields**.



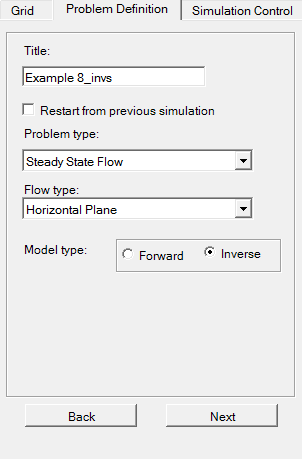
The generated field should be similar to the graph below.



**Part 2: Using well data from part one in an inverse model:**

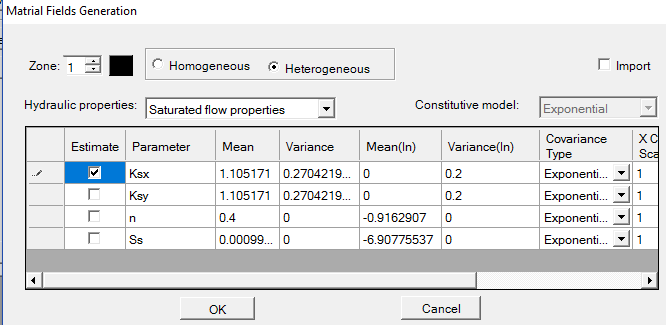
1. Problem Definition

* Select tab **Problem Definition**.
* Check **Inverse** in the model type.
* Then select the **Materials** tab.



1. Material

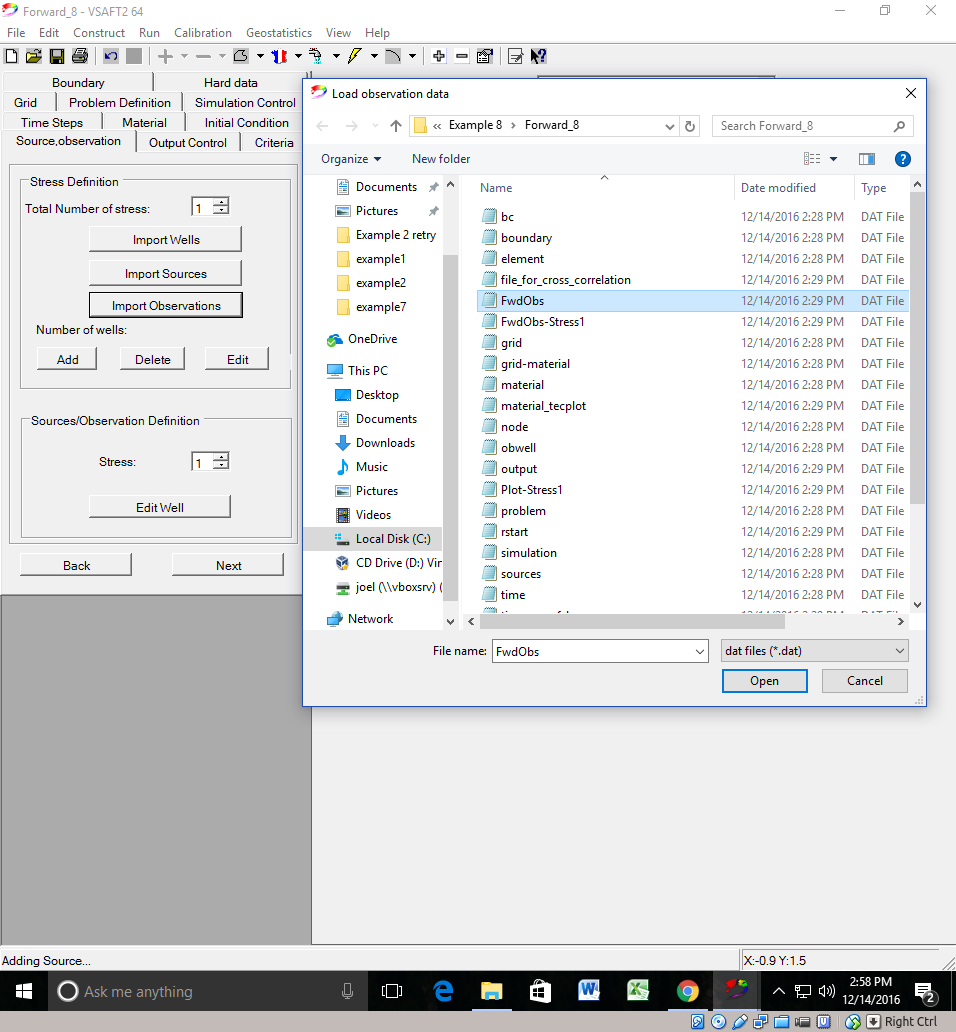
* Select the **Material** tab.
* Click **Define** in the “Zone material” properties section. A window, like the one shown below will open.
* Under the “Estimate” section select the checkbox next to Ksx. Since we are modeling a steady state saturated flow problem, we can only estimate Ksx.
* Make sure the other parameters not being estimated from the part 1 are the same (they should be).
* Select **OK.**
* Now select the **Source.observation** tab.



1. Source, observation

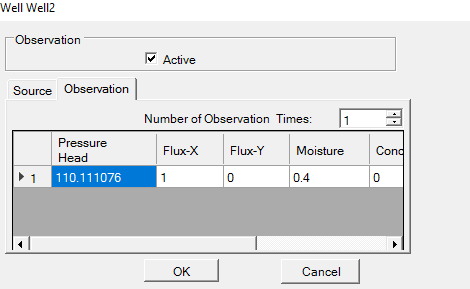
Now we can import the observed head generated by the synthetic test.

* Click **import** **observations** in the “Stress Definition” area.
* Find the location of the file **O-FwdObs.dat** located in the “Forward\_8” folder generated from part 1.
* Select **Open** to import the data.



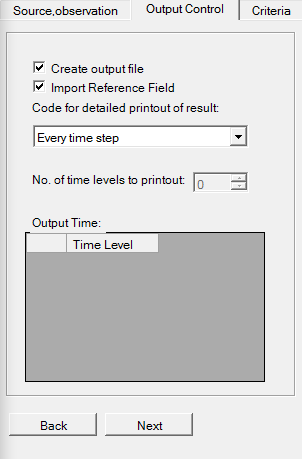
You can check whether the data is correct or not.

* Select **Edit Well** under the “Sources/Observation Definition” section and then click any of the observation well.
* A new window will open. Select the **Observation** tab and notice the fields are now populated with observational data from the forward model in Part 1.
* Select **Ok.**
* Select **Next** to continue to “Output Control” tab.



1. Output Control

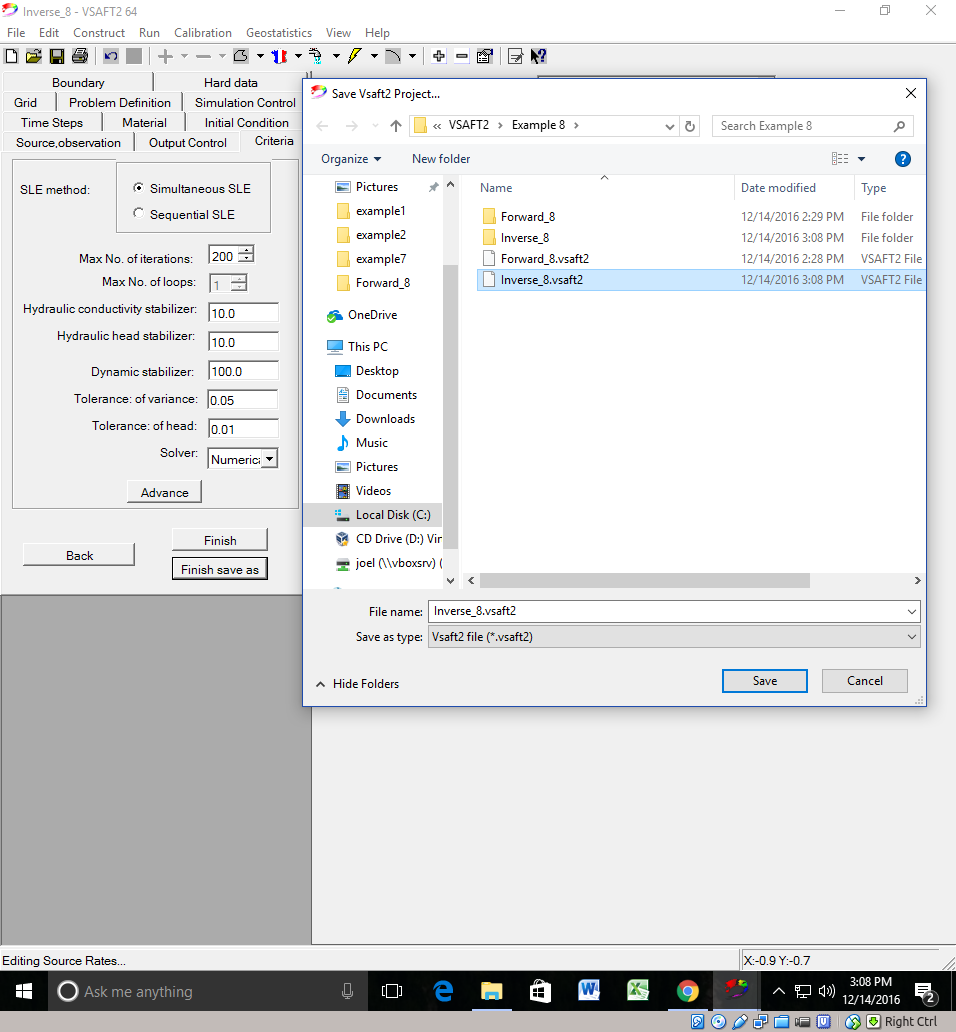
* Make sure the **Create output file** box is checked.
* Select **Import Reference Field** and a popup box will be displayed.
* Find the location of the file, **material.dat** located in the “Forward\_8” folder generated from part 1.
* Select **Open** to import the data. This will allow us to compare data from the forward problem to the results generated in the inverse problem.
* leave the other defaults
* Click **Next** to continue to the “Criteria” tab.



1. Criteria

The Criteria tab is used to define the parameters used for the parameter estimation.

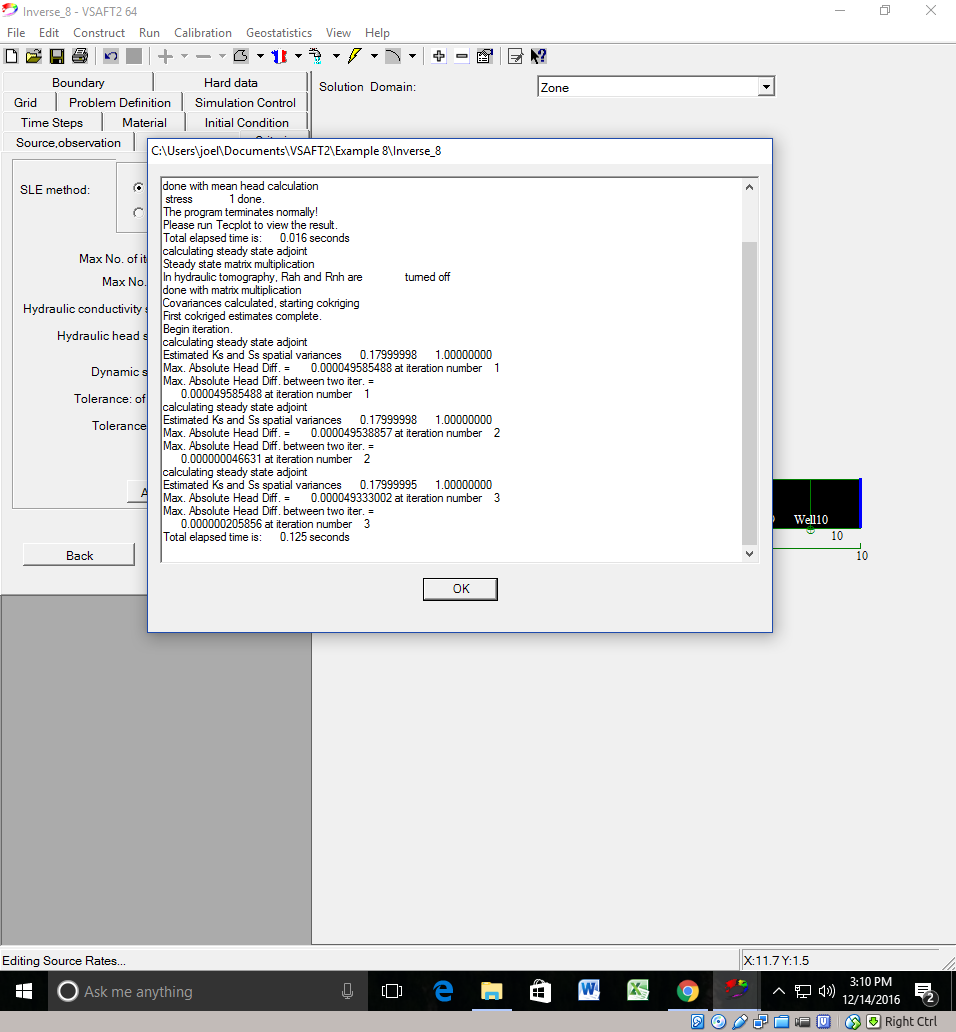
* For this example, we will keep all the parameters as default.
* Choose **Finish** **Save as**.
* Save the file as **Inverse\_8**



1. Running VASFT2

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

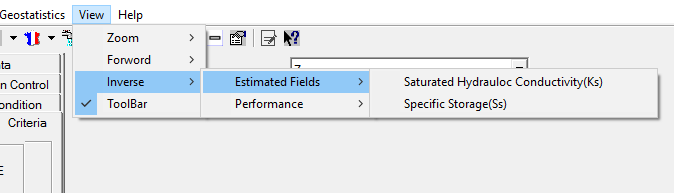
The program will run SSLE to estimate the unknown Ks values.



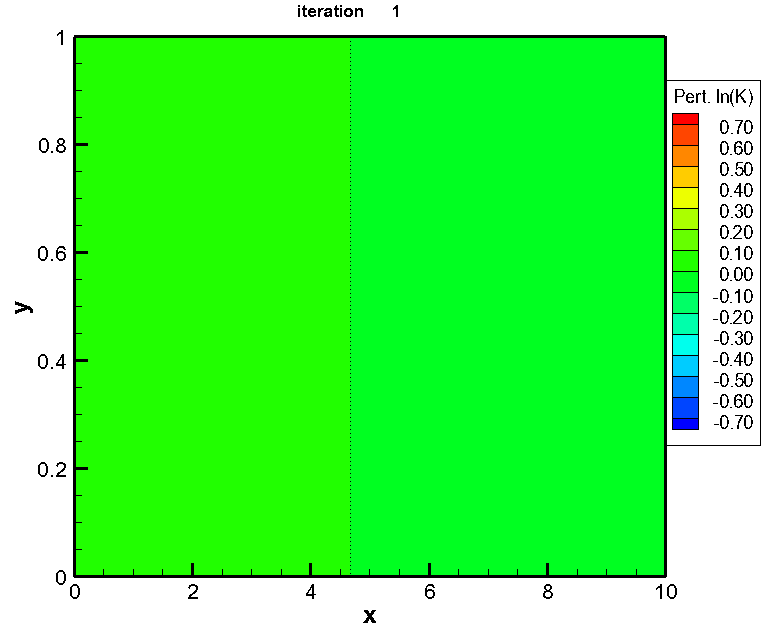
1. View results

To view the estimated hydraulic conductivity for comparison with part 1:

* Select **View->Inverse->Estimated Fields->Saturated Hydraulic Conductivity(Ks)**

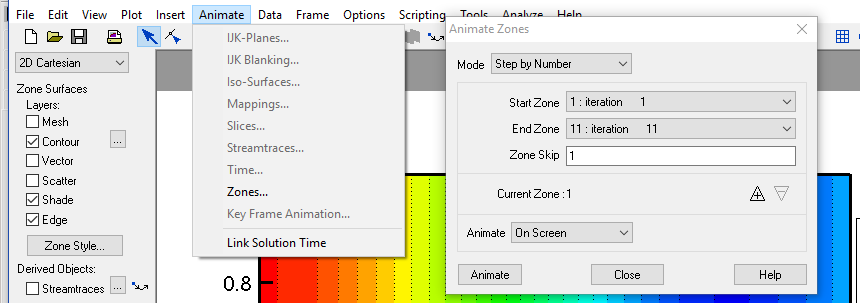


By default the plot will show the first iteration of the solver as the Pert. ln(K)



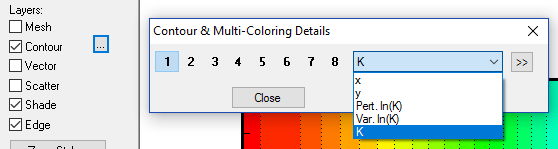
We need to advance the graph to the final iteration of the solver.

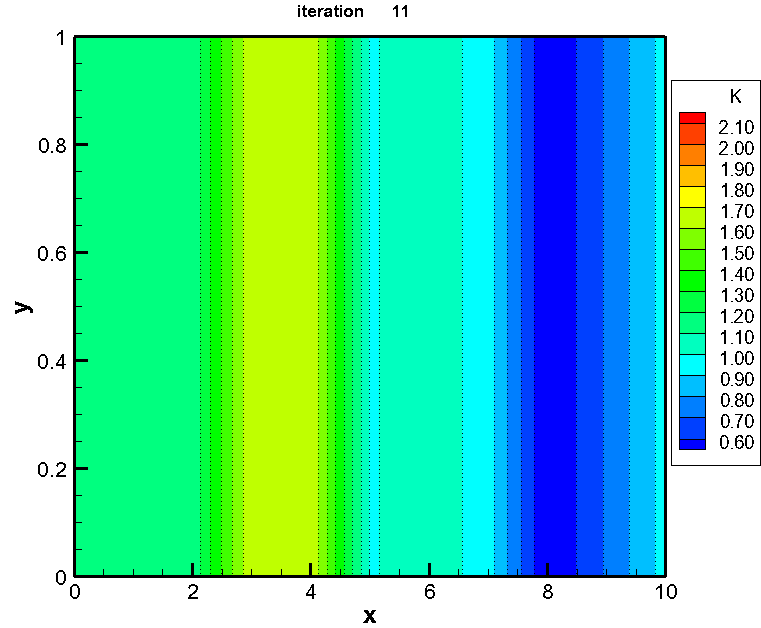
* Select **Animate->Zones**
* When the “Animate Zones” dialog box is displayed, select the **End Zone** drop down menu and select the last iteration in the list.
* Then select **Animate**.



Next we must make change ln(K) to K for comparison to our results in part one.

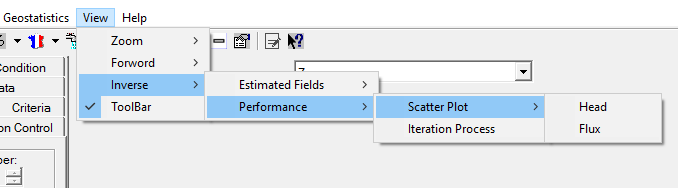
* Select the box, labeled **…** next to “Contour”
* When the “Contour & Multi-Coloring Details” dialogue box is displayed, select the dropdown box to change the variable being displayed. Select **K**.



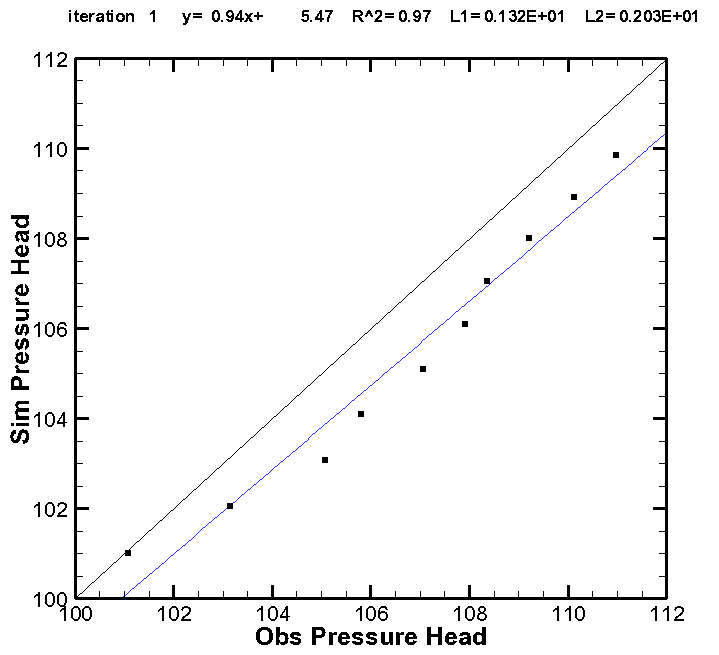
You will get a hydraulic conductivity plot very similar to part 1 as seen below. If the scale is different than the graph in part one you can adjust it manually. 

Another useful performance metric for the inverse model is a comparison of the pressure head simulated at the well sites in the domain to the pressure head observed from the forward model.

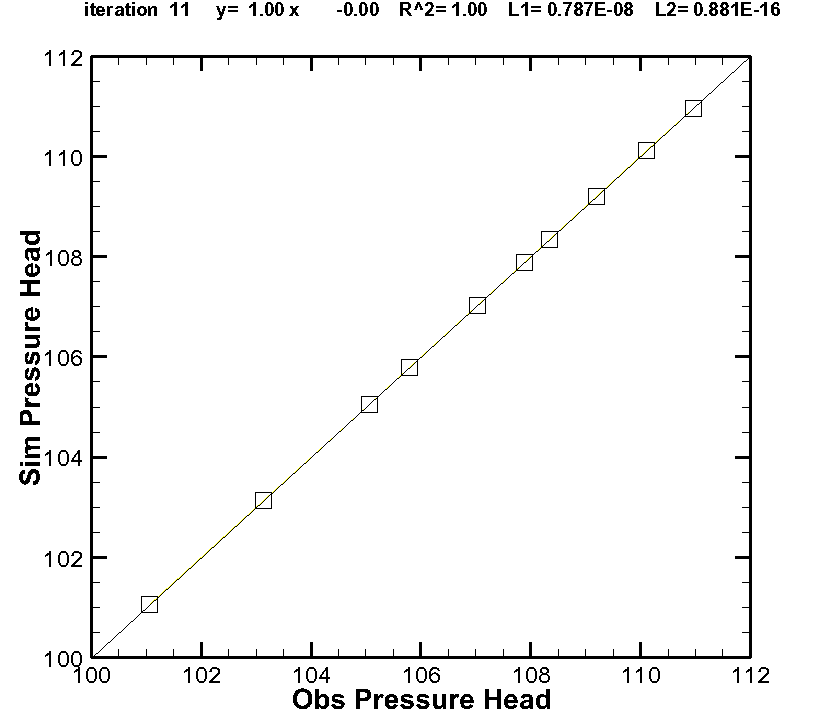
* This can be created by selecting **View->Inverse->Performance->Scatter Plot->Head**



By default the comparison of the observed pressure head to the simulated pressure head will show the first iteration of the solver. For each iteration the data solver will better fit the center black line shown below.



* To display the last iteration calculated select **Animate->Mappings**
* Once the Animate Mappings dialog box is displayed. Select the last iteration produced.

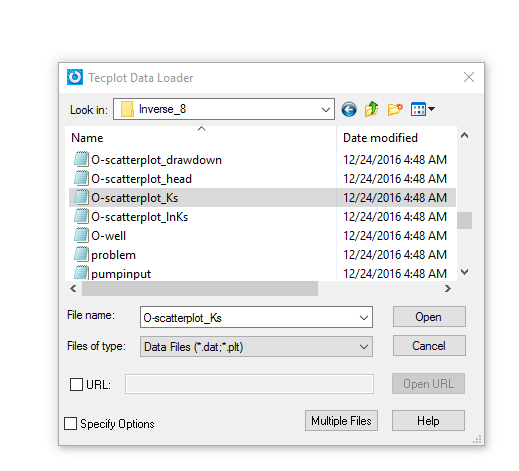
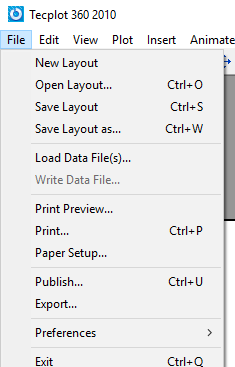


The solver finds solutions by minimizing the difference between the simulated variables and measured variables at the observation wells. Pressure head is one of the variables the program will minimize to achieve an R^2 value of 1.

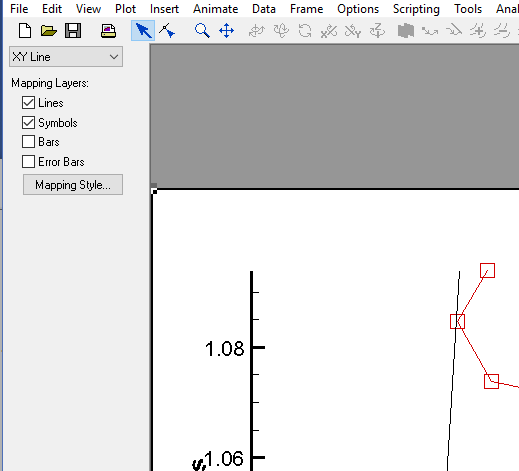
1. Comparison of K values from Forward and Inverse problem

An X-Y plot comparing the generated hydraulic conductivity values (K estimate) to the hydraulic conductivity values of the forward problem (K reference) are useful for determining the accuracy of the estimated hydraulic conductivity.

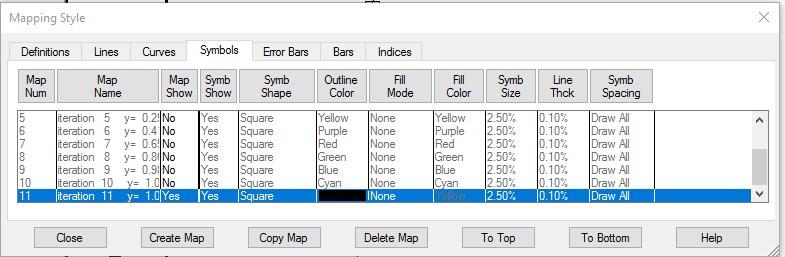
* To generate an X-Y plot open a new window of **Tecplot 360**
* Select **File->Load Data File(s)…**
* A “Select Import Format” dialog box will be displayed. Select, the default **Tecplot Data Loader** then select **OK**.
* Browse to the “Inverse\_8” folder and select **O-scatterplot\_Ks.dat** file; then select **Open**.



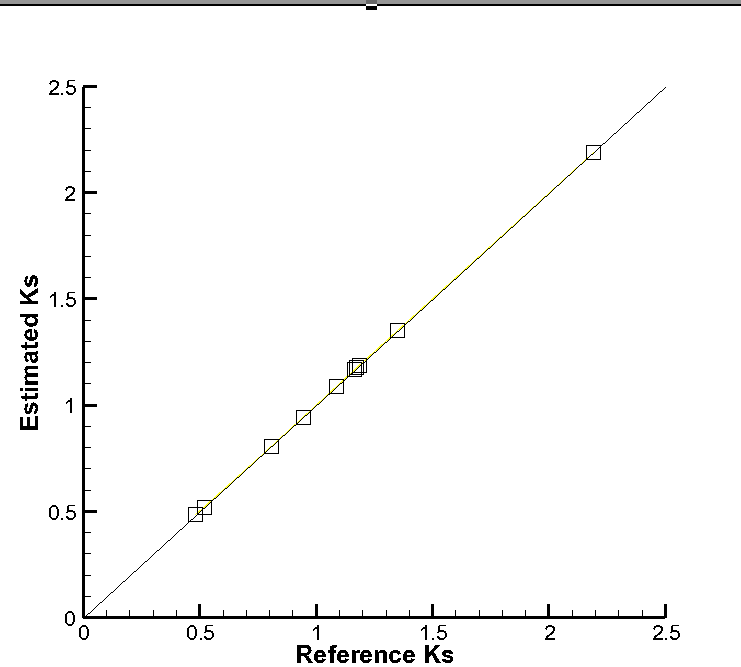
* The “Select Initial Plot” dialog box will be shown. For the “Initial Plot Type” choose **XY Line** then select **OK**.
* Select the **Symbols** check box to add symbols to every data point in the graph.
* Next click **Mapping Style** in the upper left hand corner of the “Tecplot” window to edit the axis and data being displayed.



* When the “Mapping Style” dialog box is displayed, select the **Map Show** button to change the iteration being displayed. We want to display the results of the final integration only.
* Adjust the parameters in the “Mapping Style” dialog box until your graph looks nice.



* The axes can be adjusted by selecting **Plot->Axis…** in the menu bar. Adjust both axes until all data is shown. When you are finished your graph should look something like the one below.



Because the data is well constrained, the inverse solver was able to matches the hydraulic conductivity data from the forward problem almost perfectly producing a straight line.