

# Third AnAqSim Tutorial

Building a multi-layer heterogeneous, anisotropic, and transient model with two time periods

## Contents

---

Introduction .....	2
Opening the Second Tutorial Model.....	2
Altering the Input for the Transient Simulation.....	2
General Settings for Transient Runs .....	2
Solution Settings for Initial Heads.....	2
Storage Parameter Input .....	3
Setting up Time Periods and Steps .....	3
Adding SVAS in the Single-Layer Area.....	3
Transient Input for the Well .....	4
Setting up Lines for Transient Head Profiles.....	5
Create the Initial Heads File.....	5
Solve the Model.....	5
Model Outputs and Analysis.....	6
Plots at Different Time Steps .....	6
Transient Head Profiles.....	7
Hydrograph of Well Heads.....	9
Hydrograph of River Reach Discharge .....	9
Examine Boundary Conditions at a Specific Time Step.....	10
Pathline Traced Through a Transient Field in a Longer Simulation .....	10
Hydrographs at any Point .....	13
Profiles at a Specific Time Step .....	14
Further Exploration for AnAqSim Users.....	17
Decrease the Basis Point Spacing to Improve Accuracy .....	17
Add Time Periods and Other Transient Features.....	17
Wrap up and Next Steps .....	17

## Introduction

---

This third tutorial builds on the model created in the second tutorial. The second tutorial model was steady and had a mix of single-level and multi-level regions. During this third tutorial, you will build a transient model based on the 2<sup>nd</sup> tutorial model, which serves as the initial condition for the transient models. In the transient model, the well discharge is shut off for a period of time, then turned back on at a lower discharge rate. The model may be built with either AnAqSim or AnAqSimEDU.

## Opening the Second Tutorial Model

---

Either begin with the model you had after completing the second tutorial or use the one named *tutor2.anaq* which was installed with the AnAqSim software. The folder with the tutorial input files is usually located in one of the following directories on your computer:

- Program Files/Fitts Geosolutions/AnAqSim/Documentation/tutor2.anaq (64-bit AnAqSim)
- Program Files (x86)/Fitts Geosolutions/AnAqSim/Documentation/ tutor2.anaq (32-bit AnAqSim or AnAqSimEDU)

Open the second tutorial model by selecting *File/Open* from the main menu, then locate and open the first tutorial model. Next select *File/Save As* and save this under another name such as “tutor3.anaq”.

## Altering the Input for the Transient Simulation

---

### General Settings for Transient Runs

Start by selecting *Model Input/General* and adjust it so the *Steady* box is unchecked (press *Enter* after unchecking to register the change with the database), and possibly edit the *Comments* field:

Data: general

Steady	Length_unit	Time_unit	Comments
<input type="checkbox"/>	m	d	3rd tutorial

### Solution Settings for Initial Heads

Now select *Model Input/Solution/Solve Settings* and check the table so it looks like this:

Data: solve settings

Underrelaxation	Maximum_iterations	Starting_heads_source	Almost_dry_fraction	Interface_leakage_option
0.9	20	file	0.01	<input type="checkbox"/>

All transient models need a source for starting heads and there are two options: *file* or *constant*. When *constant* is selected, the initial head is set to the value specified under *Average\_head* in the domain input. The *file* option will get initial heads from another model. AnAqSim provides a mechanism for writing heads from the other model at all locations where initial heads are needed in the transient model. Initial heads are needed at all spatially-variable area sink (SVAS) basis points and at some other locations that are used with post-solve analysis. In this model, we will use the steady 2<sup>nd</sup> tutorial model as the source of initial heads, so the *file* option is appropriate. The other settings are typical and work for most models. See the User Guide for more details.

## Storage Parameter Input

In steady models, the storativity and specific yield parameters don't factor in, so we did not pay attention to them in either the 1<sup>st</sup> or 2<sup>nd</sup> tutorials. These parameters are key in transient models and we will now edit the domains table to give reasonable input for these parameters.

Select *Model Input/Domains/Confined and/or Unconfined* and edit the *Storativity* and *Specific\_yield* data as follows:

Data: confined and/or unconfined domains

Label_unique	Level	Domain_Type	Top_elevation	Bottom_elevation	Average_head	Porosity	Storativity	Specific_yield	K1_horizor
all	1	unconfined	110	82	110	0.11	0.001	0.09	3
recharge basin	1	unconfined	110	82	110	0.11	0.001	0.09	3
3D upper	1	unconfined	110	95	110	0.11	0.001	0.09	3
3D lower	2	confined	95	80	110	0.11	0.0009	0.1	3

The three unconfined domains have specific yield = 0.09 (storativity values for unconfined domains are not used). The one confined domain has storativity= 0.0009 (specific yield values for confined domains are not used). The lower storage parameter for the confined domain is characteristic of the elastic storage of confined aquifers, compared to the larger specific yield due to water table storage in unconfined aquifers.

## Setting up Time Periods and Steps

All transient models must define the time periods and steps in the simulation. This model will have two time periods, each 2 days long with five time steps, maximums allowed in AnAqSimEDU. Select *Model Input/Time Steps* and edit the table so it looks like this:

Data: time steps

Period_length	Steps_in_period	Step_multiplier
2	5	2.0
2	5	2.0

*Step\_multiplier* makes successive steps within a period longer by the factor given. In this case, the 2<sup>nd</sup> step will be twice as long as the 1<sup>st</sup>, the 3<sup>rd</sup> step will be twice as long as the 2<sup>nd</sup>, etc. Many parameters can vary from one period to another: specified discharges, heads, stages, and recharge/leakage rates. In this simulation, the well discharge will change in each period.

## Adding SVAS in the Single-Layer Area

In the 2<sup>nd</sup> tutorial model, we used a uniform area sink (UAS) to represent recharge in the single layer parts of the model, which was fine because there was no vertical leakage and no storage changes. In this transient model, there will be small spatially-variable storage changes in the single layer areas, so SVAS are needed instead of UAS.

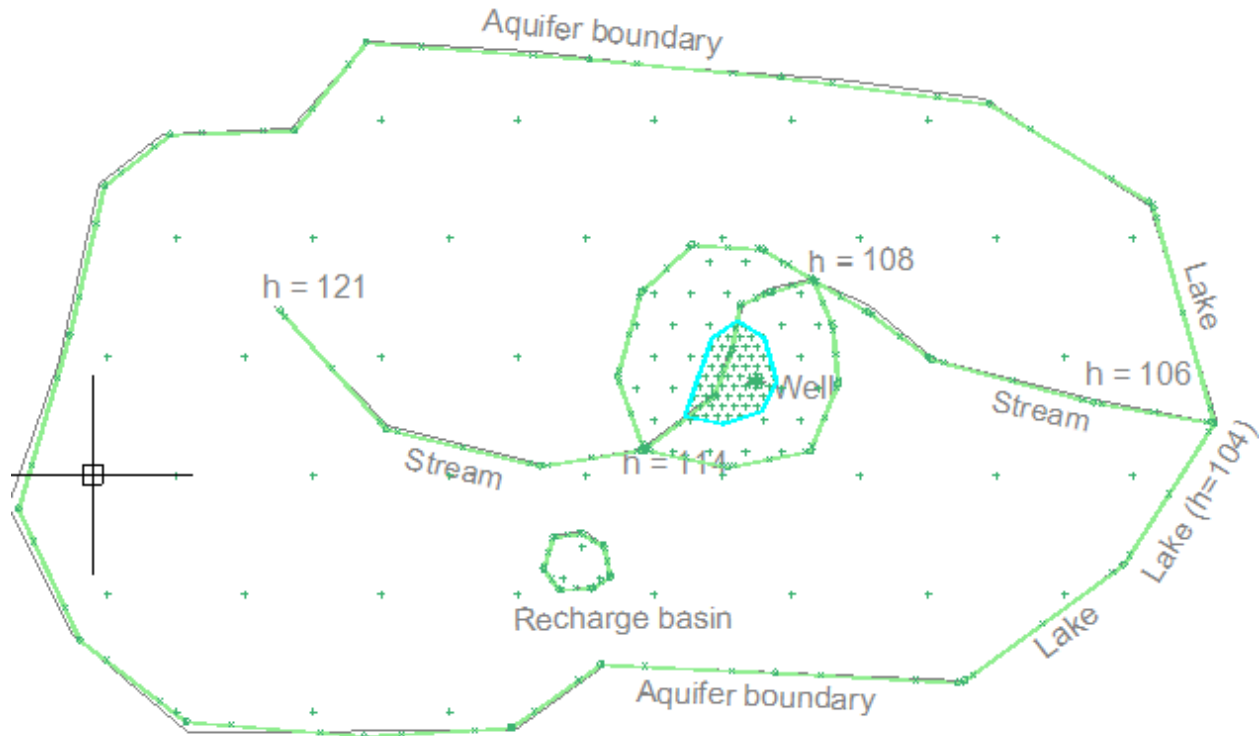
First, remove the UAS by selecting *Model Input/Area Source/Sink/Uniform, Domain* and deleting both lines of input there (highlight the rows, then press *Delete*).

Now add in SVAS to replace the deleted UAS for the "all" and "recharge basin" domains. Select *Model Input/Area Source/Sink/Spatially-Variable, Domain* and add rows for these domains so the table looks like this:

Data: spatially-variable area source/sink, by domain

Domain	Condition_Top	Top_flux_or_head	Top_surface	Condition_Bottom	Bottom_flux_or_head	Bottom_surface	Node_spacing
3D upper	Flux	0.0003	none	Flux	0	none	80
all	Flux	0.0003	none	Flux	0	none	300
recharge basin	Flux	0.008	none	Flux	0	none	80

To see how the basis points are distributed, select *Make Plot/Model Elements Only*:



With this change, spatially-variable storage changes will be modeled in all domains. The domain basis point spacings are sparse to allow AnAqSimEDU users to build this model. If you are using AnAqSim, you could decrease the basis point spacings to improve accuracy (e.g. 50 in “3D upper” and “recharge basin” and 150 in “all”, which generates about 680 equations).

Save your input with *File/Save* or *Ctrl-S*.

### Transient Input for the Well

The initial condition model is the steady 2<sup>nd</sup> tutorial model which had the well pumping at -414 m<sup>3</sup>/d.

To input the transient well discharges, move your cursor near the well in the *Plot* tab, right-click and select *Edit Nearest Well*. Enter the discharges in the *Discharge* cell like this:

Data: discharge-specified wells

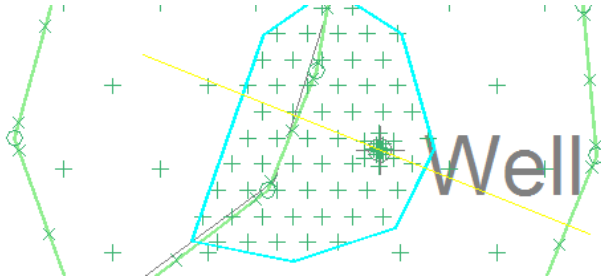
Label	Domain	X,Y	Radius	Discharge
W	3D lower	222.4071, -52.72507	0.3	0,-240

Input for transient parameters is done with a comma-delimited list as explained under *Model Input Menu* in the User Guide. The well discharge will be 0 in the first period and -240 in the second period.

## Setting up Lines for Transient Head Profiles

AnAqSim has a feature that will draw head profiles along a line that you specify. Profiles include the initial condition and each time step in the transient simulation. We will input two of them, one in each level to show the head changes near the well and river.

First, digitize a line like this yellow one that goes across the river and through the well:



Then select *Analysis Input/Transient Line Conditions* and create two lines of input like the following. For both lines, click the *Edit* button and paste in the line coordinates you just digitized.

Data: transient line conditions			
Label	Level	Coordinates	Points_on_line
Level 1	1	Edit	200
Level 2	2	Edit	200
		Edit	

Save your input with *File/Save* or *Ctrl-S*.

## Create the Initial Heads File

The initial heads for this transient run will come from the 2<sup>nd</sup> tutorial model. Follow these steps to create this file.

1. Select *File/Save Locations for Initial Transient Heads*. Use the default name, which is the same as the input file name, but ending in “.ihl”. This saves a file with all the basis point locations, well locations, and points along the transient head profile lines.
2. Select *File/Close* to close the transient model. AnAqSim gives you the option to save if you haven’t recently done so.
3. Select *File/Open*, and open the steady 2<sup>nd</sup> tutorial input file (use the same file you used to start this 3<sup>rd</sup> tutorial.)
4. Select *Solve* to solve the 2<sup>nd</sup> tutorial model.
5. Select *File/Write Initial Transient Heads* and when prompted, select the “.ihl” file you saved in step 1. This will save the initial heads in a binary file with the same name, but “.hds” at the end.
6. Select *File/Close* to close the initial model.
7. Select *File/Open*, and open the transient 3<sup>rd</sup> tutorial model.

## Solve the Model

Now begin solving the transient model by selecting *Solve*. When prompted for the initial heads file, select the .hds file you created in the previous step. The system of equations is solved at each time step, and like the

steady model, there are nonlinearities that require iteration to home in on the solution. We have two time periods of 5 steps each, so there are 10 time steps. The information written to the *Log* tab for this model will look something like this:

```
Log | Start solving system 8/20/2016 3:58:26 PM
    | 482 equations and unknowns in system
    | Time Period 1, Time Step 1, Iteration 1 2 3 4 5
    | Converged within the specified solution check values.
    | Time Period 1, Time Step 2, Iteration 1 2 3 4
    | Converged within the specified solution check values.
    | Time Period 1, Time Step 3, Iteration 1 2 3 4
    | Converged within the specified solution check values.
    | Time Period 1, Time Step 4, Iteration 1 2 3 4
    | Converged within the specified solution check values.
    | Time Period 1, Time Step 5, Iteration 1 2 3
    | Converged within the specified solution check values.
    | Time Period 2, Time Step 1, Iteration 1 2 3 4
    | Converged within the specified solution check values.
    | Time Period 2, Time Step 2, Iteration 1 2 3 4
    | Converged within the specified solution check values.
    | Time Period 2, Time Step 3, Iteration 1 2 3
    | Converged within the specified solution check values.
    | Time Period 2, Time Step 4, Iteration 1 2 3
    | Converged within the specified solution check values.
    | Time Period 2, Time Step 5, Iteration 1 2 3
    | Converged within the specified solution check values.
    | Solve complete 8/20/2016 3:58:33 PM
```

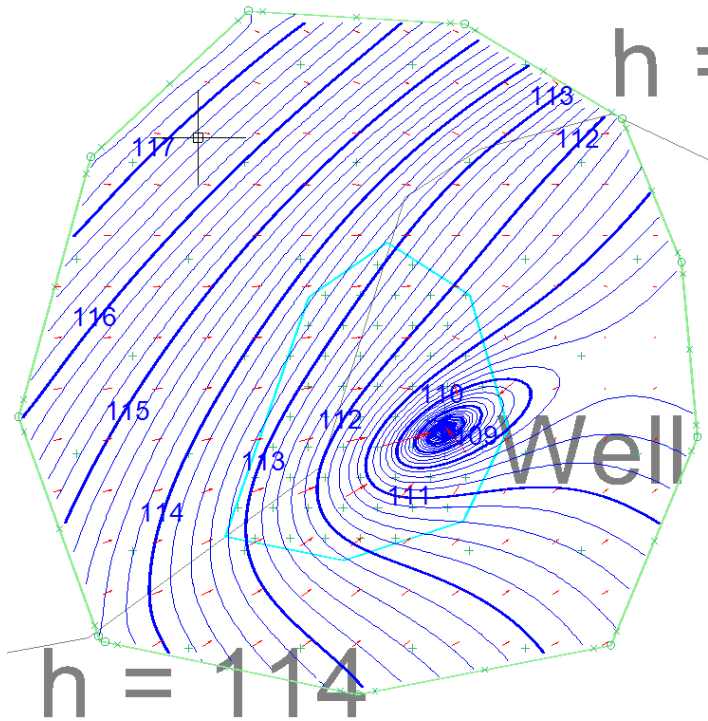
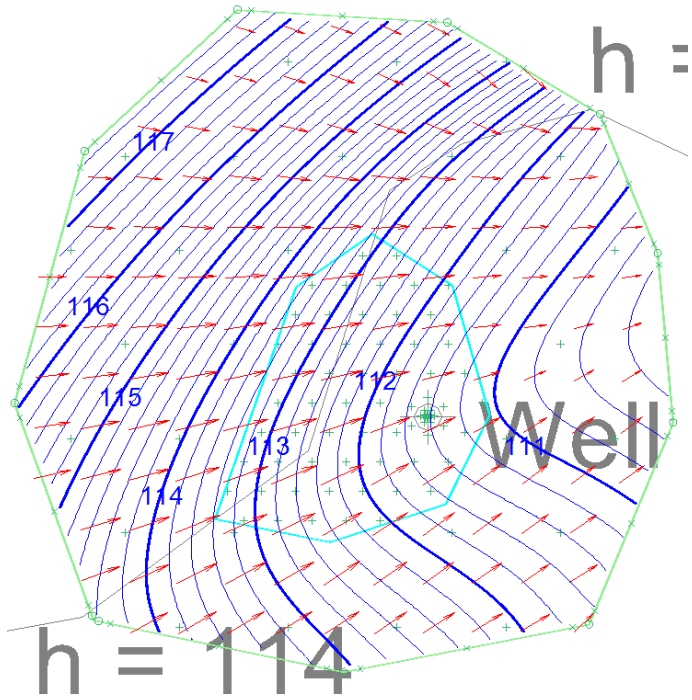
Fewer iterations are needed at later time steps, since the system changes less from one step to the next towards the end of a period.

## Model Outputs and Analysis

---

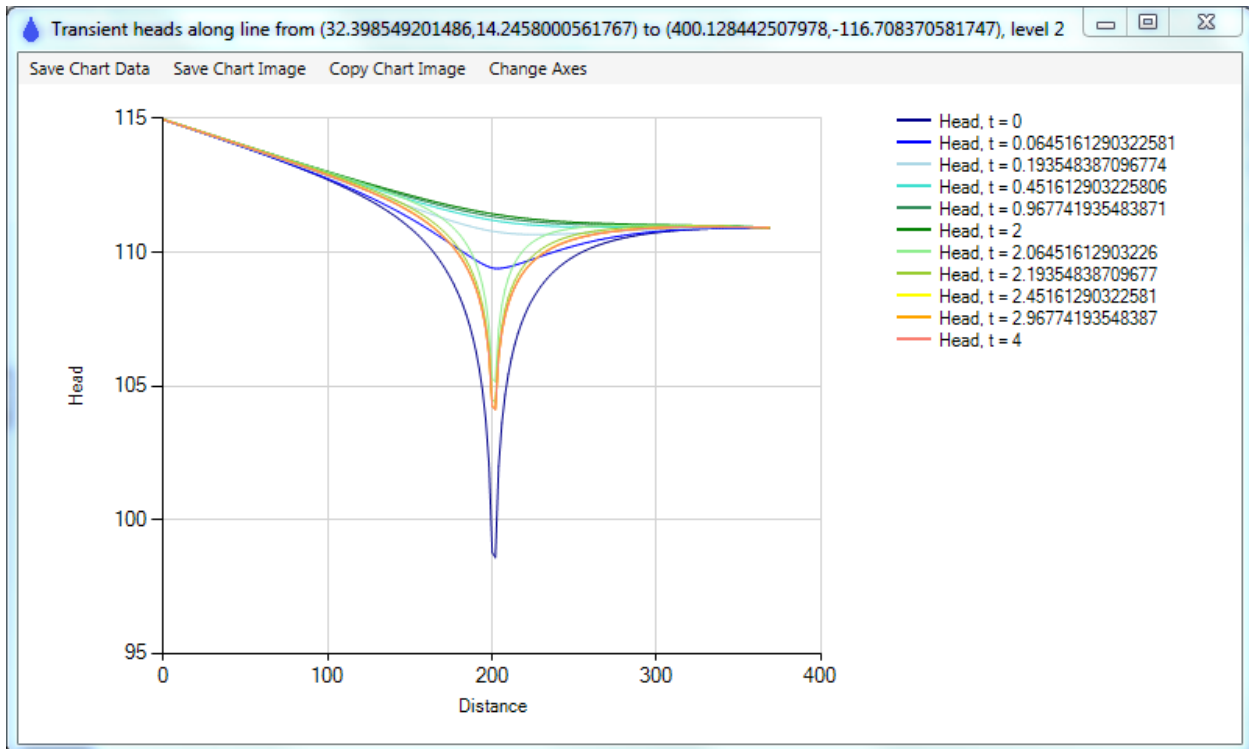
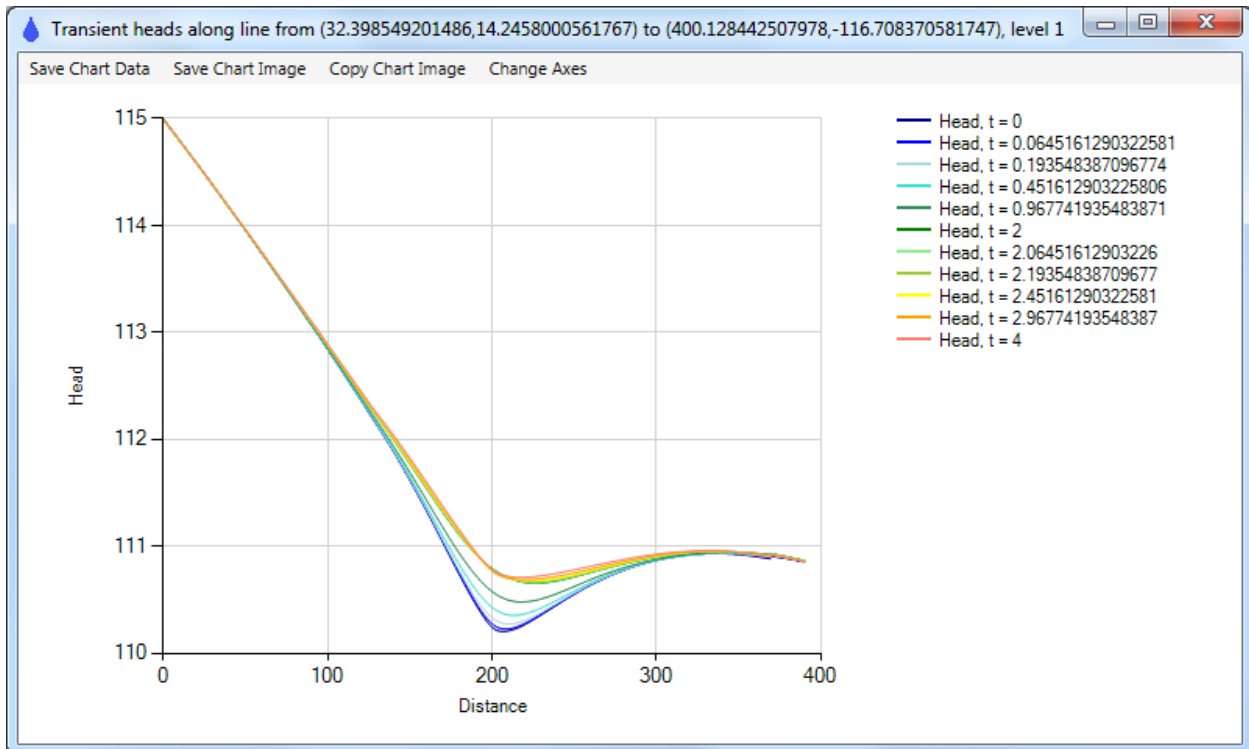
### Plots at Different Time Steps

In transient models, you can make plan-view plots of contours, vectors, etc. at any time step in the simulation by selecting *Make Plot/All Selected Features, Choose Level/Time*, then choosing the level and time period. Below are plots of head contours and vectors in level 2 at the end of period 1 (well has been off) and at the end of period 2 (well has been on at a reduced rate), just as an example. Try making a few plots of your own at different time steps.



### Transient Head Profiles

Now create transient head profiles along the lines we set up previously. Select *Analysis/Graph Transient Head Along a Line* and two graphs will be generated for the two lines we input under *Analysis Input/Transient Line Conditions*. The profiles in level 1 and 2, respectively will look like the following two graphs.

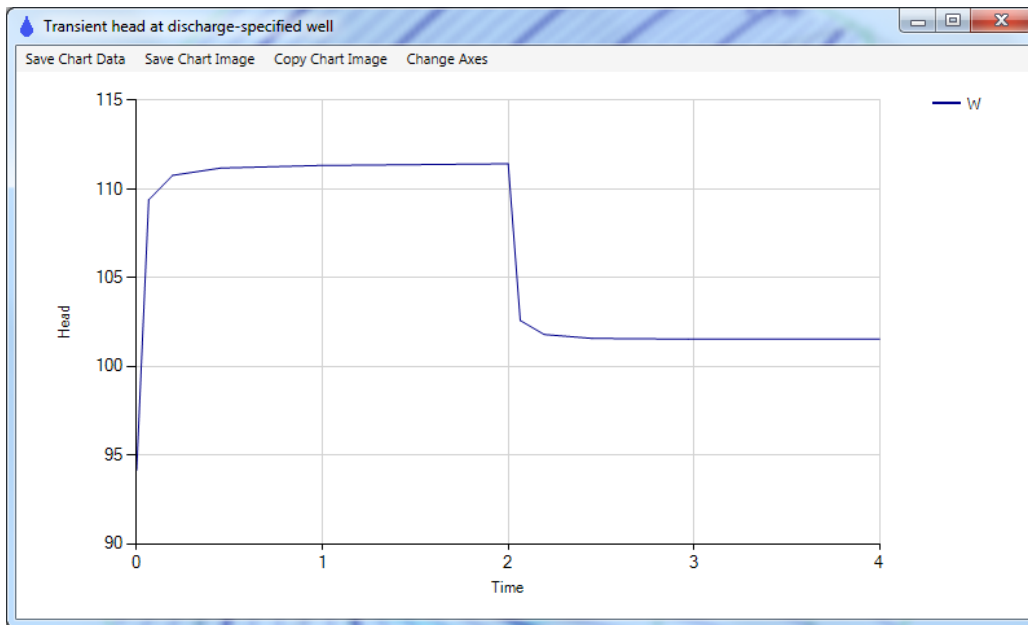


The level 2 profile shows the heads near the well screen rebounding about 14 ft during the first period when discharge = 0, and then drawing down about 8 ft during the second period when the discharge = -240. Response in level 2 is rapid and large compared to level 1, in part because the storativity of the confined domain "3D lower" is much smaller than the specific yield of the unconfined domain "3D upper".



## Hydrograph of Well Heads

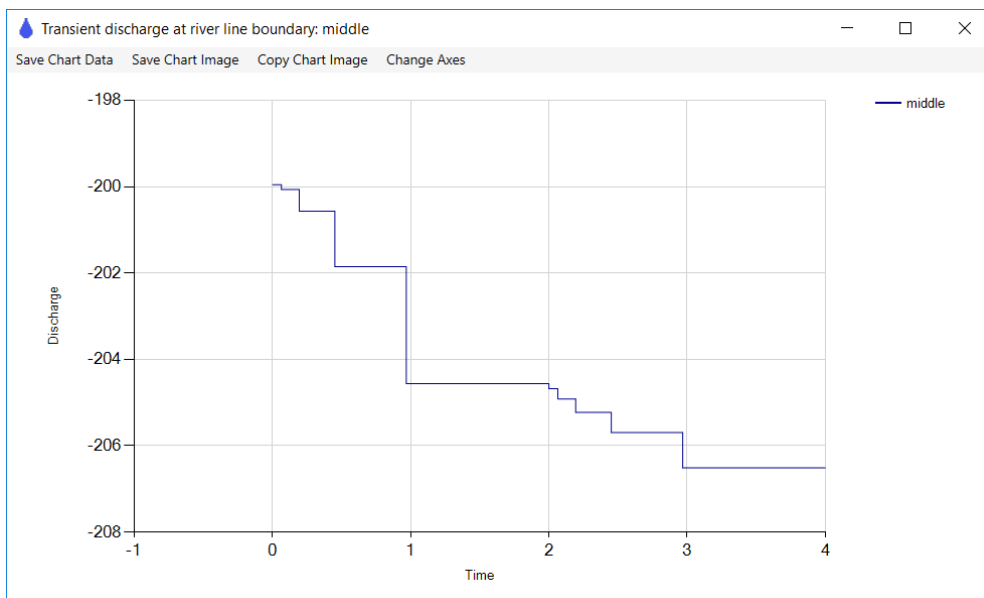
To make a head hydrograph for the pumping well during the simulation, Select *Analysis/Discharge-Specified Well Heads/Graph Transient Heads* and then select the one well. The resulting plot will look like this:



This shows the head at the well radius vs. time in the simulation. Heads become nearly stable fairly quickly, and the head during period 2 is not as low as the initial head because the period discharge is -240 compared to -414 in the steady initial model.

## Hydrograph of River Reach Discharge

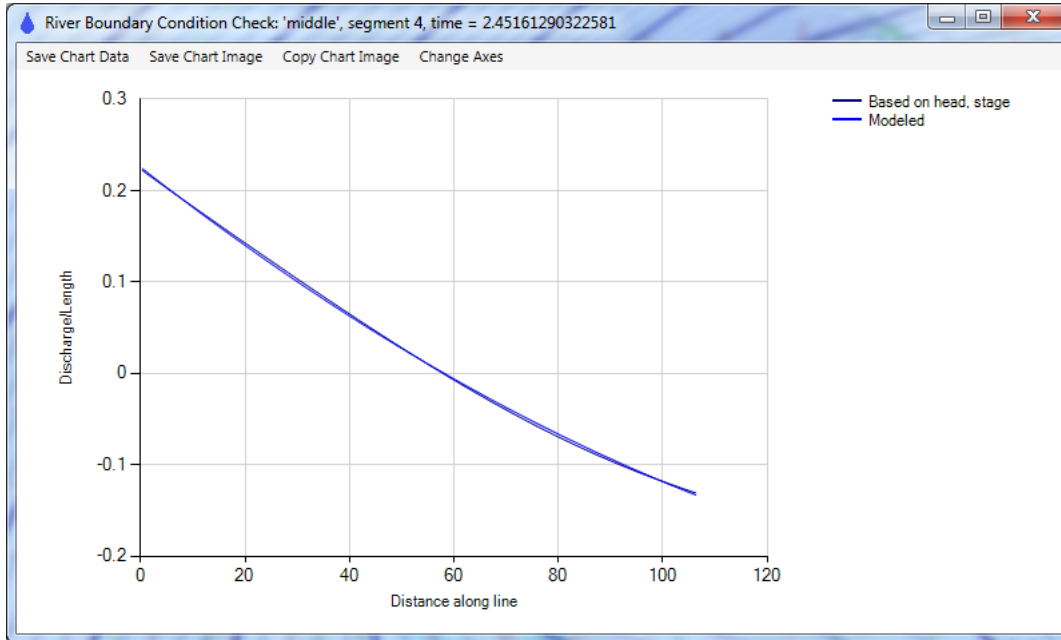
There is a similar option for generating a discharge hydrograph for river reaches. Select *Analysis/River Line Boundary Discharges/Graph Transient Discharges* and then select the middle river reach to create a plot like the following.



This reach is near the well and when the well turns off and heads rise, discharge to the river increases (becomes a larger negative number). It is still increasing during the 2<sup>nd</sup> period because heads are still rebounding some near this reach. Note that discharges are constant through each time step.

### Examine Boundary Conditions at a Specific Time Step

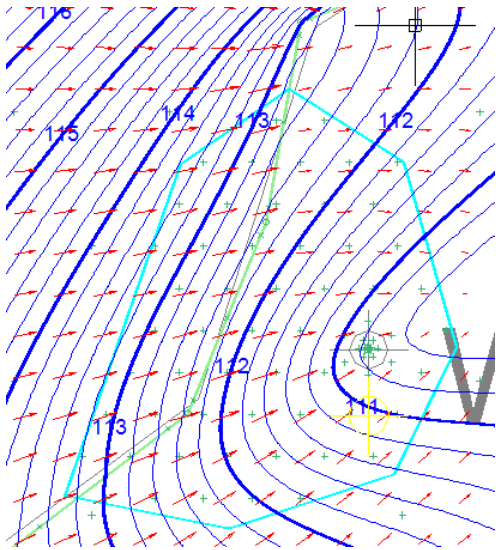
Just like in steady models, you can check the boundary condition at segments of line boundaries, but in this case, the condition is checked at the specific time step that is shown in the plot. After making a plot of any time step, move your cursor near the river segment that lies directly west of the well, right-click and select Check Nearest Line Boundary Condition. Here is a resulting plot for the 3<sup>rd</sup> time step of the 2<sup>nd</sup> period (t = 2.45 d):



Experiment with checking other boundary conditions at other time steps.

### Pathline Traced Through a Transient Field in a Longer Simulation

Pathlines in transient models are traced through the transient flow field. We will trace a single pathline from a point to demonstrate the impact of the changing well discharge. First, digitize a point just south of the well by right-clicking, then selecting Digitize/Point, and pick a spot similar to the yellow point in the following:



Then select *Plot Input/Pathlines/Single* and complete the row as follows, pasting in the digitized point into the X,Y column.

Data: single pathlines

Label	Direction	X,Y	Start_level	Show
Level 2	downstream	222.639262647896, -86.6853103704973	2.5	<input checked="" type="checkbox"/>
				<input type="checkbox"/>

The pathline will be traced downstream from mid-height in level 2, the same level where the well is screened. To make the pathlines reasonably long, make the simulation longer by selecting *Model Input/Time Steps* and editing the *Period\_lengths* so they are both 50 days instead of just 2 days:

Data: time steps

Period_length	Steps_in_period	Step_multiplier
50	5	2.0
50	5	2.0

Now, select *Solve* to solve this new, longer duration model.

Next we will adjust the pathlines settings. Select *Plot Input/Pathline Settings* and edit the table as follows.

Data: pathline settings

Step_size	Time_markers	Time_marker_incre	Total_time	Start_time	Freeze_tran	Capture_coi
0.005	<input checked="" type="checkbox"/>	10	20000	0.0	<input type="checkbox"/>	<input type="checkbox"/>

The pathline will start at t=0, the start of the simulation, and run for the full transient simulation time of 100 days. The *Total\_time* of 20000 is greater than the simulation time, so pathlines will only be traced for the duration of the simulation. Arrow markers will be drawn at 10 day intervals along the pathlines.

Now edit the line and well pathline tables, making sure to un-check the *Show* column of all pathline input in those tables:

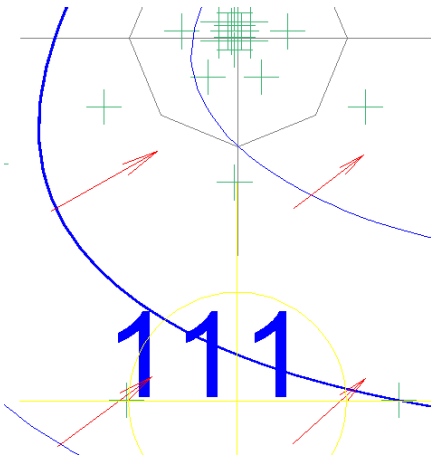
Data: line pathlines

Label	Direction	Coordinates	Number	Start_level	Show
upstream of recharge	downstream	Edit	10	1.0	<input type="checkbox"/>
		Edit			<input type="checkbox"/>

Data: well pathlines

Well Label	Number	Start_level	Show
W	15	1.2	<input type="checkbox"/>
			<input type="checkbox"/>

In the *Plot* tab, zoom into a close-up view of the single pathline point like the following, and then right-click and select *Set Plot Window to Current View*, so the plot will be a close-up of the pathline area.

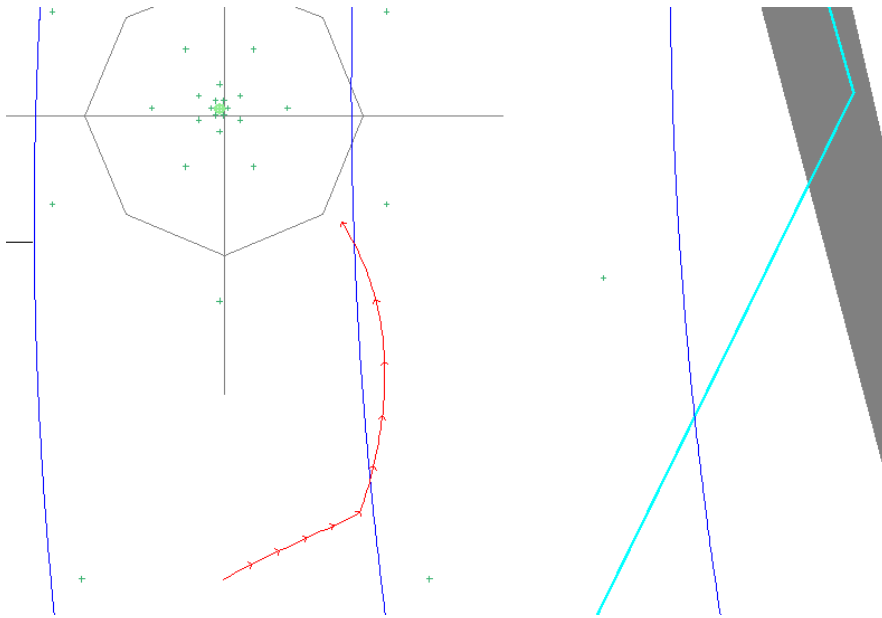


Select *Plot Input/What to Plot* and un-check *Vectors* and check *Pathlines* as follows. Make sure to press *Enter* after making the change, to register the change with the database.

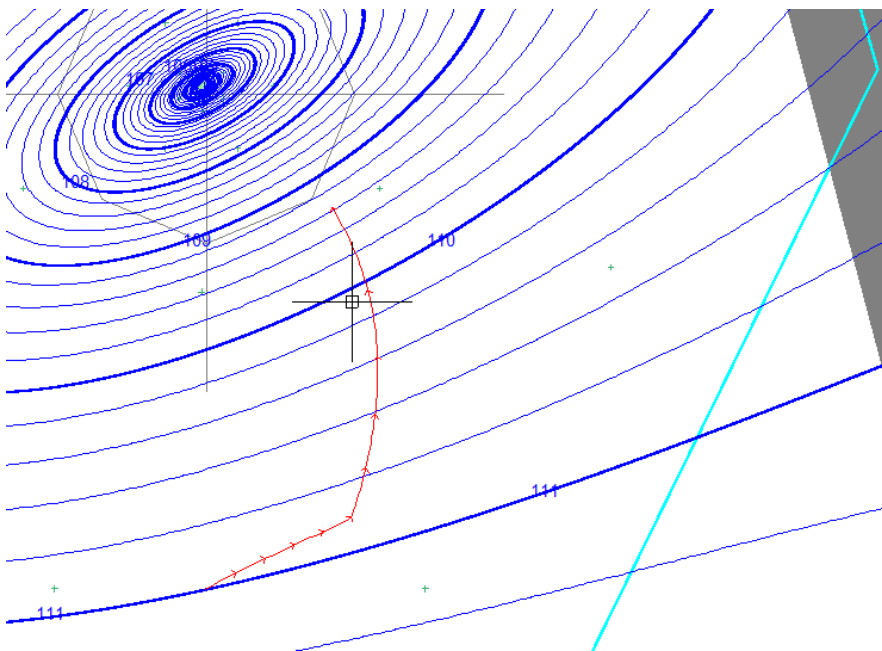
Data: what to plot

Window	Basemap	Basemap_File	Elements	SVAS_polygons	Contours	Pathlines	Vectors	Head_Calibrat
199.09562	gray	Select	elements w/ details, con	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Now make a plot. Select *Make Plot/All Selected Features* and choose level 2 at t=50, the end of the 1<sup>st</sup> period. You will see a plot like this:



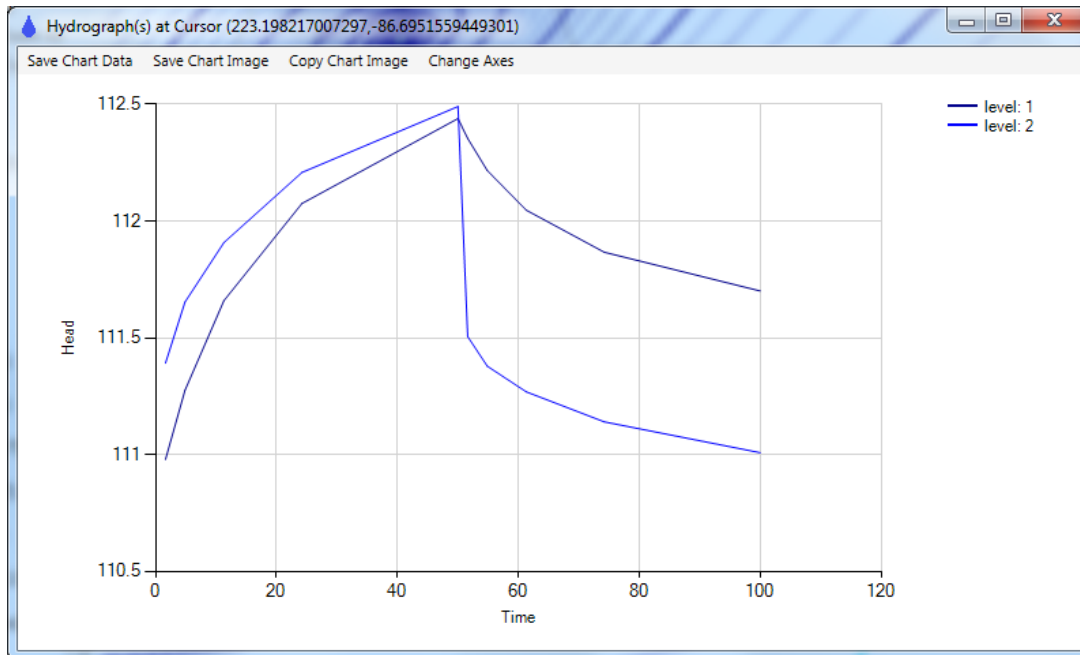
Repeat, but choose level 2 at  $t=100$ , the end of the 2<sup>nd</sup> period to get a plot like this:



The pathlines that are displayed travel through time in the simulation, whereas the contours drawn are for a specific time during the simulation. Move the cursor over the arrowheads and you will see that the pathlines turn abruptly when the well begins discharging again at  $t=50$  days. The contours show that during period 1, the head gradient is more towards the east and less steep than during period 2, when the well discharge induces a steeper northward gradient. Remember that the horizontal  $K$  is anisotropic, so the pathlines do not generally migrate normal to the head contours like they would if  $K$  was isotropic.

### Hydrographs at any Point

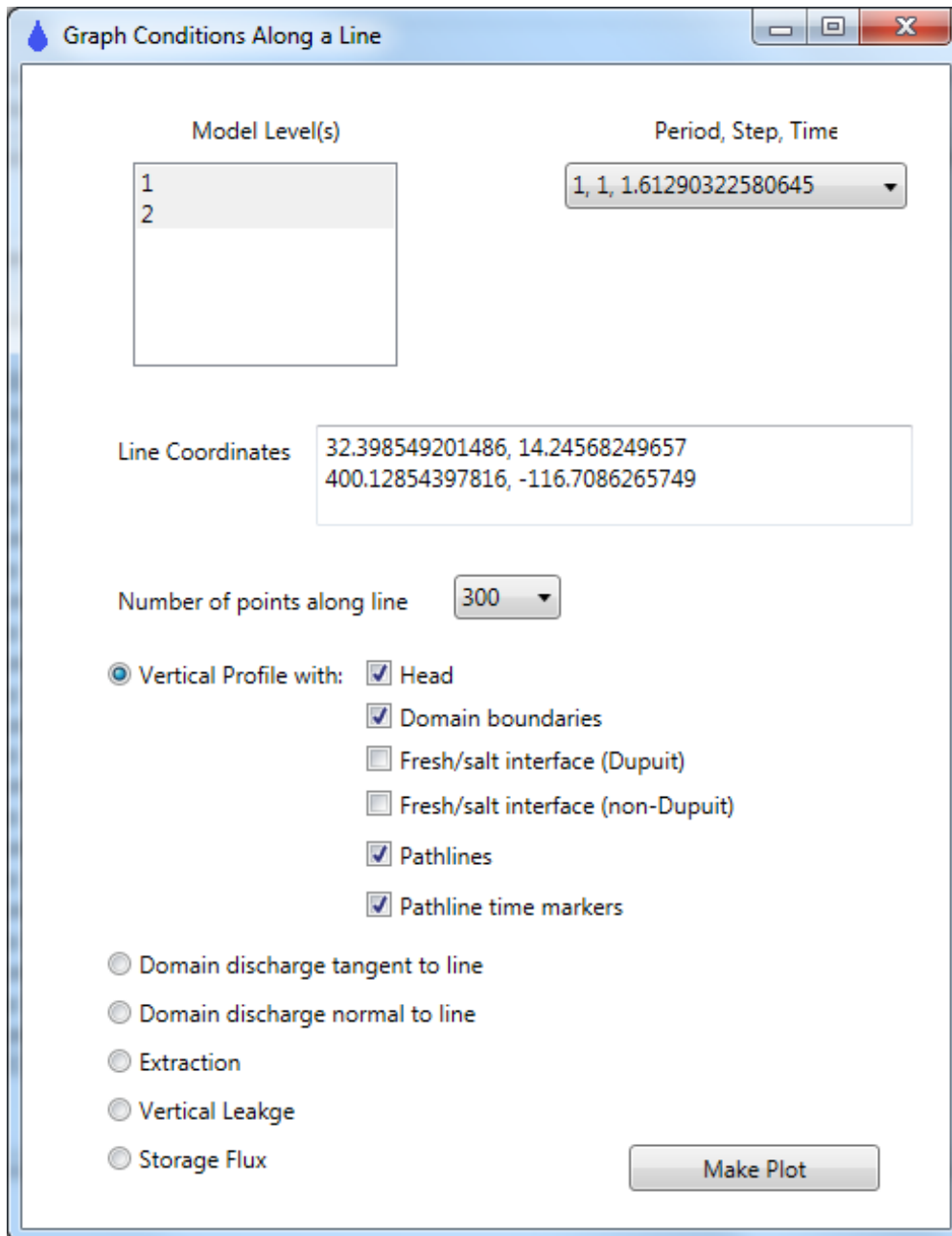
You can make head hydrographs at any point in the model by moving the cursor to a desired location, right-clicking and selecting *Graph Head Hydrograph(s) Here, All Levels*. The following are hydrographs of this 100-day simulation near the starting point of the pathline.



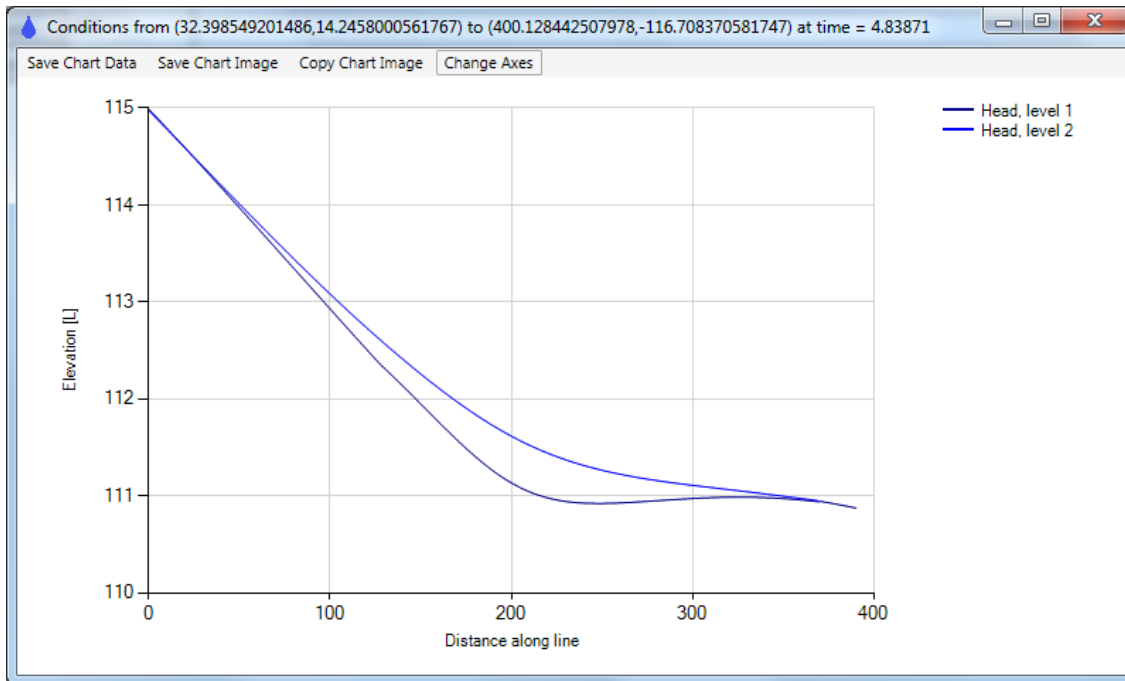
In period 1, heads rise after the well was shut off, and heads in level 1 are lower than in level 2 because there is upward flow as heads rebound from the lack of pumping in level 2. In period 2, heads draw down due to pumping, especially in level 2 where the well is screened. You can see that the response is quicker in confined level 2 than in unconfined level 1.

### Profiles at a Specific Time Step

You can make profiles of head, domain boundaries, pathlines, discharge, and extraction at any time step in the simulation. We will examine profiles of head and extraction. We will use the same profile line that was input for our transient head profiles. Select *Analysis Input/Transient Line Conditions*, click one of the *Edit* buttons under *Coordinates*, and copy the coordinates there. Then select *Analysis/Graph Conditions Along a Line*, paste these coordinates into the *Line Coordinates* box, select both levels, the second time step, select *Vertical Profile*, and check *Head*:

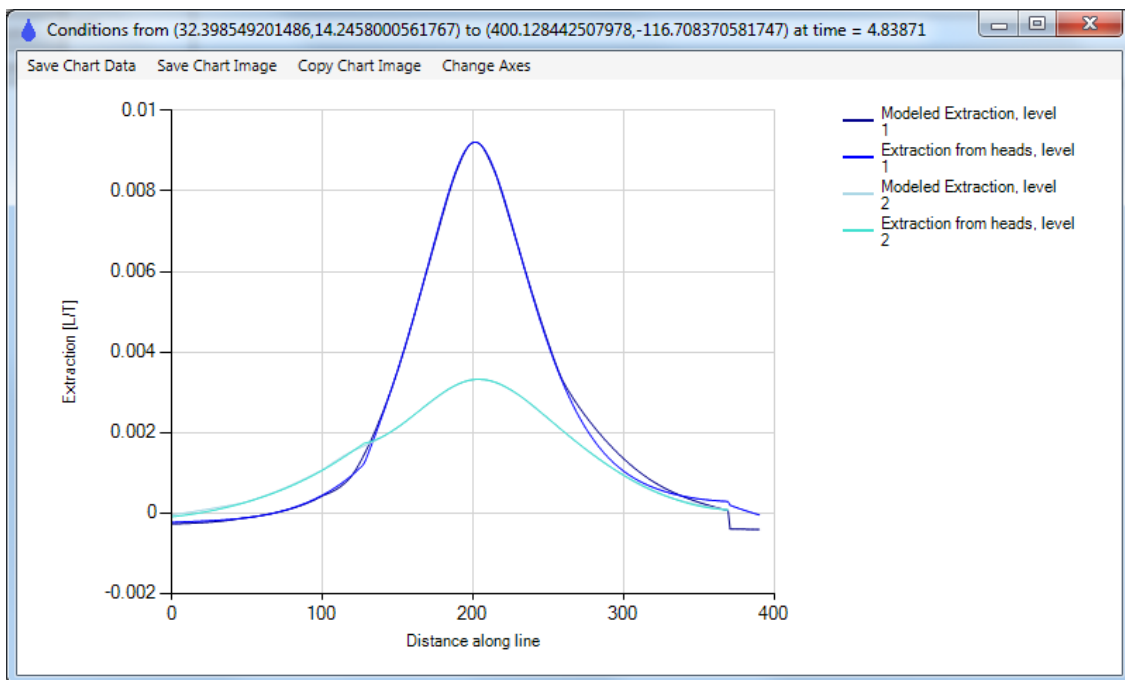


When you press *Make Plot*, you will get a head profile like this, which shows head rebounding after the well discharge changes to zero:



Head rebounds more quickly to higher levels in level 2 (confined, low storativity) than in level 1 (unconfined, high specific yield).

Now select *Analysis/Graph Conditions Along a Line* and choose *Extraction* and click on *Make Plot* to generate this plot of extraction:



This shows generally good agreement with the modeled extraction close to the heads-based extraction, but you can see significant deviation in level 1 near the right side of the profile, where the basis point spacing is large. This can be cured by decreasing the basis point spacing, which is something you can explore if you are using AnAqSim. Extraction near the well is positive in both levels because heads are rising and water is going into storage.



Save your model at this point.

## Further Exploration for AnAqSim Users

---

This tutorial model was near the 500 equation limit for AnAqSimEDU, but AnAqSim users could modify and expand this model in many ways. A few of the possibilities are outlined below.

### Decrease the Basis Point Spacing to Improve Accuracy

At the end of the previous section, we saw that outside the polygon SVAS, the accuracy of modeled extraction was only fair. Modify the model by reducing the basis point spacing in the “all” and “3D upper” domain SVAS. After doing that, solve the model and check the extraction using *Analysis/Graph Conditions Along a Line* like was done in the previous section. You should see significant improvement in the extraction match.

### Add Time Periods and Other Transient Features

AnAqSim users can add any number of time periods and steps. If you add more periods, you can have key inputs vary in each period. Here are some ideas for other transient inputs to try:

- Add a head-specified line boundary (*Model Input/Line Boundaries/Head-Specified*) and/or head-specified wells (*Model Input/Pumping Wells/Head-Specified*) to simulate a dewatering system. You could move the head lower in stages in different periods. Using the *Off\_Periods* input, you can shut down the system for certain periods. You may want to input denser basis point spacing in the vicinity of the dewatering system. Examine the discharge of the line boundary or wells through time using *Analysis/Internal Head-Specified Line Boundary Discharges/Graph Transient Discharges* or *Analysis/Head-Specified Well Discharges/Graph Transient Discharges*.
- Alter the recharge rates for different time periods. Edit the domain and polygon SVAS input to add different rates at different times. Check the simulation results using hydrographs and/or transient head profiles.
- Alter the river stages in different time periods. Check the river reach discharges vs. time with *Analysis/River Line Boundary Discharges/Graph Transient Discharges*.

## Wrap up and Next Steps

---

This concludes the third tutorial. If you want to see the input file that was generated by this tutorial or any of the tutorials, you can find the files in the same Documentation directory where the basemap was located:

- Program Files/Fitts Geosolutions/AnAqSim/Documentation/tutor3.anaq (64-bit AnAqSim)
- Program Files (x86)/Fitts Geosolutions/AnAqSim/Documentation/ tutor3.anaq (32-bit AnAqSim or AnAqSimEDU)

To learn more about AnAqSim, please explore the [checks of AnAqSim](#), which include model input files for the check models.