

AnAqSim Instructional Series

Hello World – A Simple Model

Ex_2-1

Hello World – Building A Simple Groundwater Flow Model

Introduction

This is a “Hello World” type introduction to model building in AnAqSim. It is very simple and can be completed in about 15 minutes.

We will build a simple model of drawdown around a pumping well in a portion of an aquifer defined by a circular boundary located at some radial distance from the well. We will set the water level at the boundary to 100 feet to create a 100-foot thick aquifer, pump the well in the center, and contour drawdown to examine the result.

Installation Instructions: AnAqSim

To complete the “Hello World” exercise you must install AnAqSim. A free demo version is available from Yellow Sub’s AnAqSim website. To install the free version of AnAqSim follow the link to the home page on the AnAqSim website (<https://anaqsim.com/>).

1. Click on the “Download AnAqSim” button. Enter the required information and click the “Free Download” button. This will cause the system to automatically send you an email with download instructions.
2. Click on the download link in the email you receive. This will download a Microsoft Software Installer (.msi) setup file.
3. Install the program. It should take no more than a few minutes to complete the installation.
4. The 64 bit version of AnAqSim should be easily accessible in the C:\Programs\Yellow Sub Hydro folder, and is also accessible in the Program List on your Start Menu.

Before You Begin

The instructions below are presented as a sequence of actions such as:

Model Input / General

This means go to the **Menu** bar at the top of the window, click on the **Model Input** menu, and then click on **General** from the list of menu items to open the data entry grid for that item.

Some operations are performed using commands in the AnAqSim **Plot View Menu**, such as:

Plot View Menu > Digitize / Point

Click **OK**.

This means go to the **Plot View Menu** at the top left of the Plot View window, click on the **Digitize** tool, select **Point**, and click the **OK** button to dismiss the tool information dialog before proceeding with digitizing.

All the files that you will need for this exercise are located within the “Ex_2-1” folder. When the instructions say to open a file, it will be located directly within this folder. In the “Completed Steps” folder, you can find model files that have already been completed for certain steps in the exercise. These can help you troubleshoot your own model files or move forward in the exercise, even if you encounter a problem. We recommend that you save all the files that you produce during this exercise in the folder called “Ex_2-1 Work” that can be found inside the “Ex_2-1” folder.

What We Will Learn In This Exercise

- How to define a model domain and set its properties.
- How to create a boundary for the domain.
- How to create a pumping well in the aquifer.
- How to set plotting and contouring parameters.
- How to solve the model and plot head contours.
- How to annotate the model output map.

1

Double click the AnAqSim icon.

Click the **Maximize** button in upper right corner of the AnAqSim window, if necessary, to expand the AnAqSim software window to fill the computer screen.

- **Launch the program.**

2**Model Input / General**

You should see the **General** data grid to enter data.

Set the following options.

Steady:	Checked
Length_unit:	feet
Time_unit:	days
Comments:	Hello World Example

Enter**File / Save**

Use filename “hello_world” (without quotes); save this file a folder called “Ex_2-1 Work”.

 **Note:** It is important to press the Enter key whenever indicated to preserve the input data.

 **Note:** Pressing Tab after entering a data value moves the cursor to the next field.

- **Fill in General model data.**
- **Any units can be entered. These are simply text strings used by AnAqSim to report certain model input values and output results, so any convenient text can be used (e.g. you can use either “feet” or “ft”).**
- **Model parameter values entered for the model must be specified in the selected units (no conversion is performed by the software).**

3

Plot Input / What to Plot

Window:	-10000, -10000, 10000, 10000
Basemap:	none
Basemap_file:	[Leave as is — empty]
Elements:	Click in cell; select “elements w/ details, control points”
Contours	Checked
Pathlines	Unchecked
Vectors	Unchecked
SVAS_Polygons	Unchecked
AreaPathline_Polygons	Unchecked
Calibration	Unchecked
Analysis_Inputs	Unchecked

Enter**File / Save**

- **Set a model space window from lower left corner (-10000, -10000) to upper right corner (10000, 10000). AnAqSim will plot all calculated results using this window.**
- **Selecting “elements w/ details” will allow us to inspect certain element properties using the cursor inspector tool once the model is constructed.**
- **Checking “Contours” turns on contour plotting once we have solved the model.**

4

Plot Input / Contour Settings

Surface:	h
Subtract:	no
Points_Evaluated:	40000
Increment:	0.5
Minimum:	[Leave as is — empty]
Maximum:	[Leave as is — empty]

Enter**File / Save**

- **Select 40000 evaluation points. This means that the model solution will be evaluated at 40000 points throughout the model domain, which provides very high resolution for contouring.**
- **Set the contour interval or increment to 0.5 ft.**
 **Tip:** If the contour interval is left at 0 or a negative number, then the increment will be automatically set so that there will be a total of 25 contours when displayed.

Model Input / Domains / Confined and/or Unconfined

Label_unique:	aquifer
Level:	1
Domain_Type:	confined
Top_elevation:	100
Bottom_elevation:	0
Average_head:	200
Porosity:	0.1
Storativity:	0.001
Specific_yield:	0.001
K1_horizontal:	10
K2_horizontal:	10
Angle_K1_to_x_axis:	0
K3_vertical_top_half:	10
K3_vertical_bottom_half:	10

Enter

File / Save (or press Ctrl-S)

Make Plot / Model Elements Only

Accept the Level 1 setting and press **OK**.

Plot View Menu > Snap Settings

Check Snap to Grid to turn it on.

Set Snap Grid Spacing to 1000.

Plot View Menu > Digitize / Circle Polygon

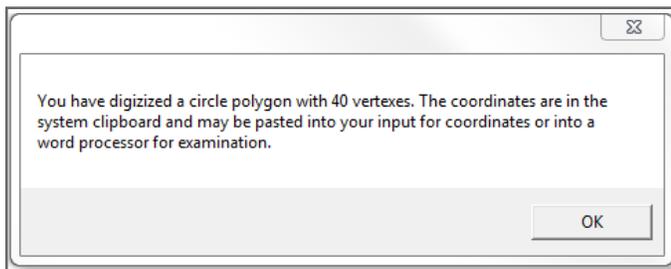
Select **40** points, and click **OK**.

Place the cursor at coordinates 0, 0.

The cursor will snap to that location.

Click and release the mouse button. Move the cursor to the right until the X coordinate reads 10000. Click the mouse button.

You should see the following dialog:



Click **OK**.

- *Use the Digitize tool to create a circular polygon for the model boundary.*
- *We will generate 40 points, evenly spaced around the circle, that will define the boundary.*
- *The digitizing tool makes sure that the first and last points in the points list are identical. This is a requirement of AnAqSim to properly close the model boundary.*

7

Model Input / Line Boundaries / Head-Specified

Click in the **Label** cell on the first data entry row. Type “boundary” (without quotes) and push **Enter**.

Enter

Click in the **Coordinates** cell. Press **Ctrl-V** to paste the circle polygon coordinates you digitized in Step 6. Click **OK**.

For the remaining data cells, enter the remaining parameter values as follows:

Domain:	Click in Cell; Select “aquifer”
Parameters_per_line:	4
Domain_Boundary:	Checked
h_start:	200
h_end:	200
Off_periods:	[Leave as is — empty]

Enter**File / Save (or press Ctrl-S)**

8

Model Input / Pumping Wells / Discharge-Specified

For the pumping well data cells, enter the parameter values as follows:

Label:	well
Domain:	Click in Cell; Select “aquifer”
X, Y:	0,0
Radius:	0.5
Discharge:	-20000

- *Place a pumping well at the center of the model — at coordinates 0, 0.*
- *Specify a 12-inch production well (radius = 0.5 ft).*
- *Withdrawal from the well is specified as negative. The discharge of 20,000 ft³/d is approximately 150,000 gpd.*

Enter**File / Save**

Your model is now ready to run.

9

Solve

Model should take a second or two to solve. It should produce output in the Run Log that looks something like the following:

```
Start solving system 8/3/20xx 11:35:02 AM
160 equations and unknowns in system
Iteration 1 2 3 4
Converged within the specified solution check values.
Solve complete 8/3/20xx 11:35:02 AM
```

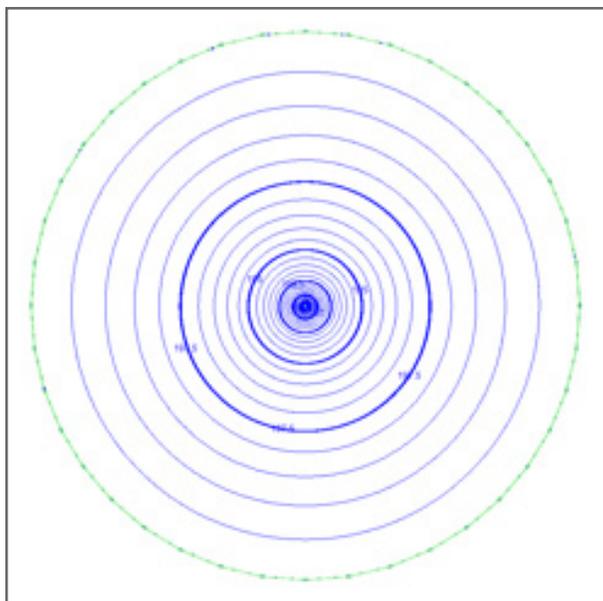
- *If the model did not converge, be sure to check that the model boundary is fully enclosed (Step 7).*

10

Make Plot / All Selected Features, Choose Level/Time / Level 1

Click **OK**.

Your results should look similar to the image below:



- *Create a plot of model results by displaying the contoured values of hydraulic head.*
- *AnAqSim uses the contour interval you specified (0.5 ft), and displays a contour label for every fifth contour line.*

Voila! Your first AnAqSim model!

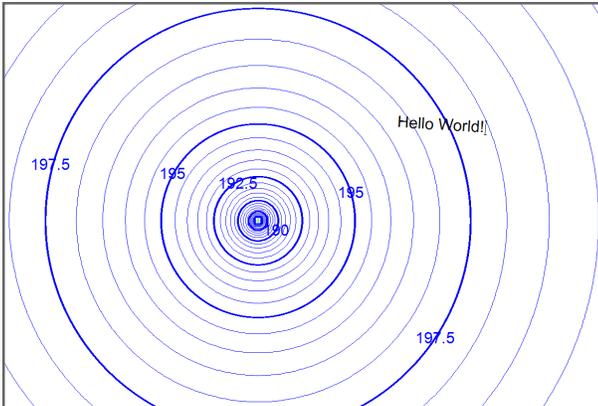
Plot View Menu > Annotate / Text

Click in the model area, and drag an inch or two horizontally to the right, and left click to start the text insertion cursor blinking.

Type “Hello World!” (without the quotes) and press **Enter**.

- **Now to make this a true “Hello World” example, add a little annotation.**

Your results should look similar to the image below.



This is just a brief example of the way AnAqSim works. Exercises in the AnAqSim training series explore how to add other hydraulic features to a model, how to create multi-layer and transient models, and how to create and save graphic outputs in a variety of digital formats.

What We Learned

- How to open a new model file in AnAqSim, set its general information, and save the file.
- How to enter a row of data to establish the properties of a domain in AnAqSim, and how to use the built-in digitizing tool to create the boundary for that domain.
- How to set up model output display parameters, including display of element information and contouring of calculated head values.
- How to solve the model, and how to check that the model solution converged.
- How to contour and display model calculated heads within the model domain.
- How to annotate a model map with text.

Additional Analyses

1. Explore model water level elevations:

Save model file “hello_world.anaq” as “hello_world_AA1.anaq.”

Solve

Make Plot / All Selected Features, Choose Level/Time / Level 1.

Move the cursor over the pumping well model domain, and look at the data output in the panel to the left of the map.

Note how X, Y coordinates change and how model heads change as you move the cursor over the model.

What is the range in heads between the well and the model boundary?

Answer: Water levels near the well are approximately 174 feet, while heads around the boundary of the model remain fixed at the specified value of 200 feet. (To get a good estimate of the calculated head at the well, zoom in before measuring.)

2. Explore model element information:

Move the cursor over the small green circles at each vertex on the boundary line element.

Note the information displayed for each of the line element vertex points showing the location of the vertex and its assigned initial head value. This provides a quick way to check the points and elements used to construct an AnAqSim model.

3. Change model boundary conditions:

Go to **Model Input / Line Boundaries / Head-Specified**, and change the specified elevation on the outer model boundary to the following values: $h_{\text{start}} = 300$, $h_{\text{end}} = 300$. Press **Enter**, and select **File / Save** to save the changes.

Go to **Model Input / Domains / Confined and/or Unconfined** and change the `Average_head` parameter value to 300. Press **Enter**, and select **File / Save** to save the changes.

It is necessary before solving an AnAqSim model to specify an average head over the domain that is close to the anticipated final solution. (For more information, search the AnAqSim User Guide for the section heading “Input Common to All Domains” and read about the Average head setting.)

Go to **Solve** to rerun the model. Go to **Make Plot / All Selected Features, Choose Level/Time / Level 1** to display the contour results.

Examine the contours; move the cursor over the model domain and inspect the head values. How are the new model head values different from the original model (see the model heads in Step 10 above)?

Answer: Water levels near the well are approximately 278 feet, while heads around the boundary of the model remain fixed at the specified value of 300 feet. You can see that the shape and depth of the drawdown cone are very similar to the previous model, although (as should be expected) drawdown at the well is slightly less for this modified model because the saturated thickness – and thus the transmissivity – are greater. In general all calculated model heads are 100 feet greater because of the 100 foot increase in the boundary condition.