# PLANNING WATER SUPPLY BOREFIELD WITH AMWELLS

This lesson explains **how to design a water supply wellfield** in an environmentally sensitive area using the AMWELLS module. After completion of this lesson, you will be able to use AMWELLS for analytical modelling. You will also be able to illustrate your results by plotting hydrographs, piezometric and drawdown contour maps and profiles.

### *Conceptual model*

A vegetable farm located in the Perth region of Western Australia requires additional water supply of 500 $m^3$/day over the next 10 years. Sufficient water resources are available in a surficial aquifer composed by alluvial sands and silts. The aquifer has a thickness of 50 m, and it is underlain by low-permeable shales. Hydraulic tests resulted in a hydraulic conductivity of 0.4 m/day and a specific yield of 0.1. The aquifer static level in the borefield area is 5 m below the ground level.

There is an environmentally protected area in the western part of the project, where vegetation relies on shallow water table (Figure 1). The environmental requirement is that drawdown at the border of this zone cannot exceed 1 m during the entire period of the wellfield operation. For this lesson, we will assume that discharge areas (i.e. water expression at the ground surface, where water evaporates) are sufficiently far from the borefield and do not impact its operation.

Figure 1 shows the site aerial photo with the outlined protected zone boundary and proposed water supply wells.

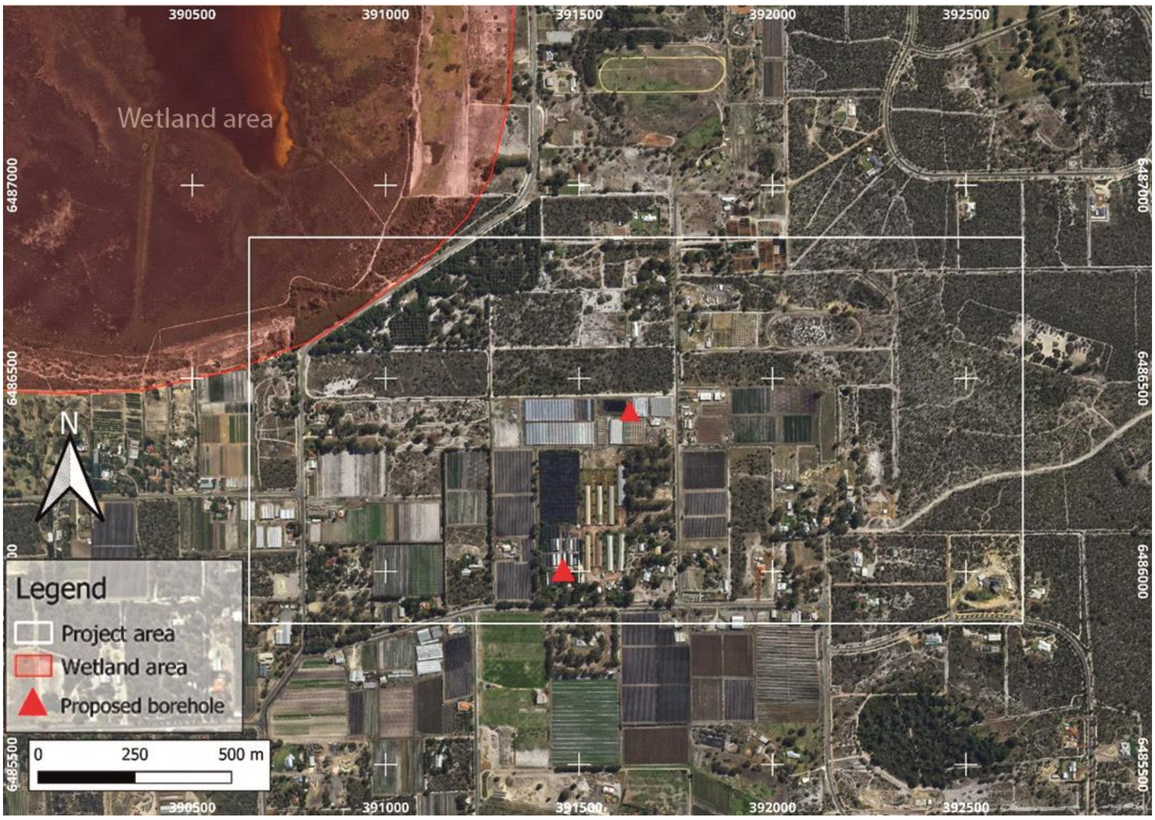



**Figure 1**. Aerial photo showing the project area, the proposed borehole locations and the protected zone.

***Lesson workflow***

The proposed wellfield consists of the two boreholes which locations are presented on Figure 1. The two boreholes will be fully penetrating the unconfined sandy aquifer. This borefield should be supplying 500 $m^3$/d of water over the 10-year period under the following conditions:

- Water level in all pumping wells shall not drop below 5 m above the base of the aquifer (operational condition). This means that drawdown should not exceed 40 m (i.e. 45 m of initial saturated thickness with subtracted 5 m of the minimum saturated thickness);
- Drawdown at the protected zone border shall not exceed 1 m (environmental licencing condition).

The following workflow will be applied for this Lesson:

1. Create a model, select a conceptual scheme and assign hydraulic parameters;
2. Import and georeference a map to the model;
3. Input proposed pumping wells, their pumping rates, time duration and aquifer properties;
4. Calculate drawdown in each well;
5. Calculate drawdown at the border of the protected area;
6. Decide if the proposed wellfield satisfies requirements;
7. If needed, optimize by varying the number of wells, their locations and their rates.

***Create a new model and define its dimensions***

To create a new project, use the command *Flow model/Analytical model/New model*. This opens the "Create analytical model" window (Figure 2). This dialog window prompts you to select conceptual model, model dimensions and other critical parameters for the model.

For this lesson, the aquifer is unconfined and consists of one infinite isotropic and homogeneous layer. The layer has a thickness of 50 m with a static water level of 45 m (i.e. 5 m below the ground level). Wells are fully penetrating and the model has the dimensions of the study area on Figure 1:

- 2000 m long in the E-W direction,
- 1000 m wide in the N-S direction.

The coordinates for the point of origin are E:390647 and N:6485865; the reference of the coordinate system used is EPSG 7850 (MGA 2022, zone 50).

The coordinate origin of the model is located in the bottom left corner of the project area. All locations are defined as distances in meters from the model origin along each axis. Entering the coordinates of the origin allows you to use the same spatial coordinates as in your GIS project. By default, the origin coordinates are (0,0): that can be changed later as well.

Once your parameters have been assigned, your screen should look like Figure 2. Click on "*Create*" to generate a *.*atm* project file and open the analytical model. Call the file "*Lesson6*".

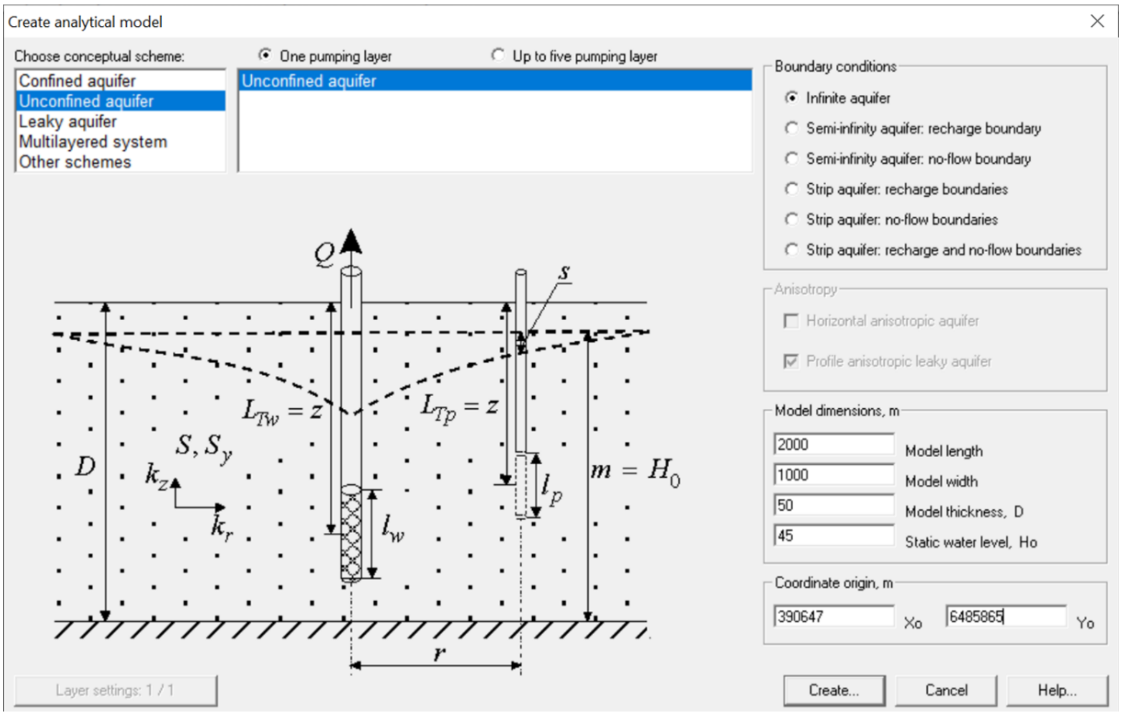



**Figure 2**. Conceptual options to create a new analytical model.

Once the file is created, the analytical model window opens. It should look like Figure 3.

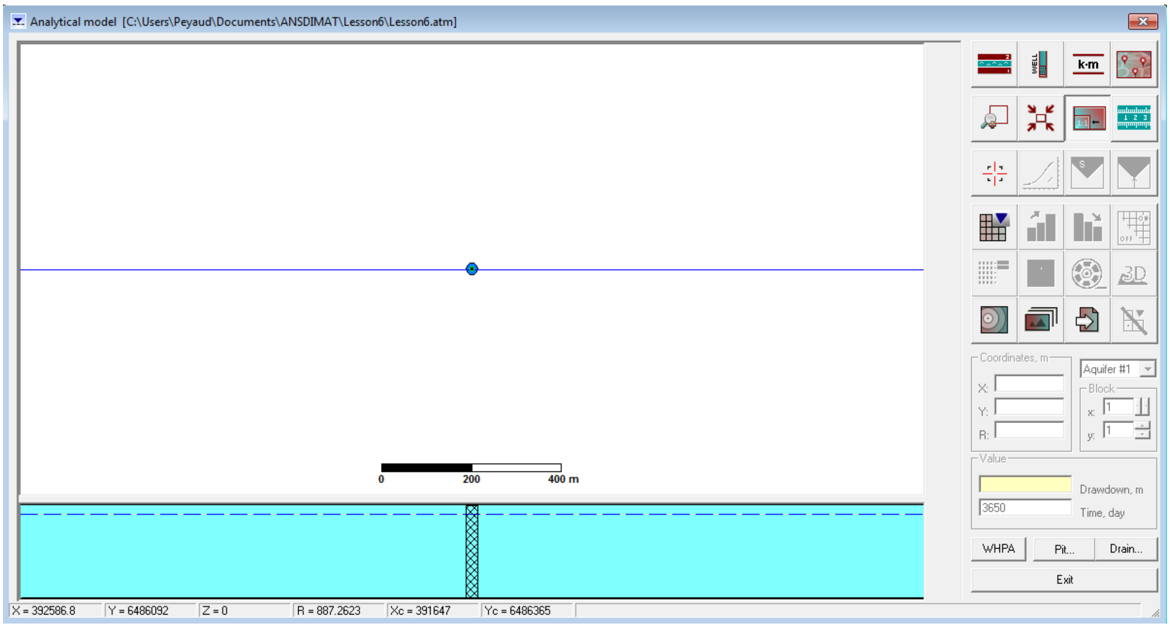
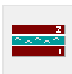



**Figure 3**. Analytical model window.

The "*Model size*" button  allows checking and modifying the dimensions of the model. Open the Model dimensions window and check that all your parameters are correct. The window should look like Figure 4.

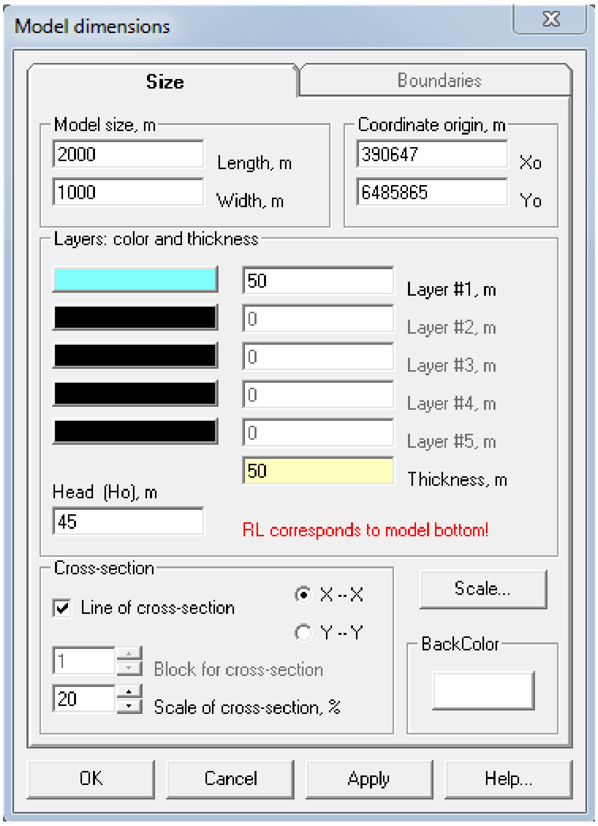



**Figure 4**. Model dimensions dialog window, where the dimensions of the model can be modified.

***Importing map to the model***

When the model opens, its graphical interface window displays a blank map view showing the extent of the model area (Figure 3). On this view, a double dot marks the location of a default pumping well, located in the centre of the area and a blue line indicating the location of the cross section. The cross section itself is located below the map showing the layer, the static water level and a well.

The status bar is located below the cross section and it displays coordinates of the mouse pointer (X, Y, Z), the coordinates of the selected well (Xc, Yc) and either the distance to the centre of the well (R when the mouse pointer is over the map area) or the saturated thickness (s) when the mouse pointer hovers above the cross section.

To import a map into the model, click on the "Load a picture" icon , then browse to the location of the file you want to add (Figure 5). ANSDIMAT does not currently support .tif files but supports a number of other graphical formats. You can prepare an image of your prospective borefield using any GIS software and export it to a supported format, for example, jpg. To appear on the correct area of the model, the picture must cover the exact same area as the model. In this Lesson the map was created in QGIS by extracting the study area of 2000 by 1000 m from the coordinates of the point of origin.

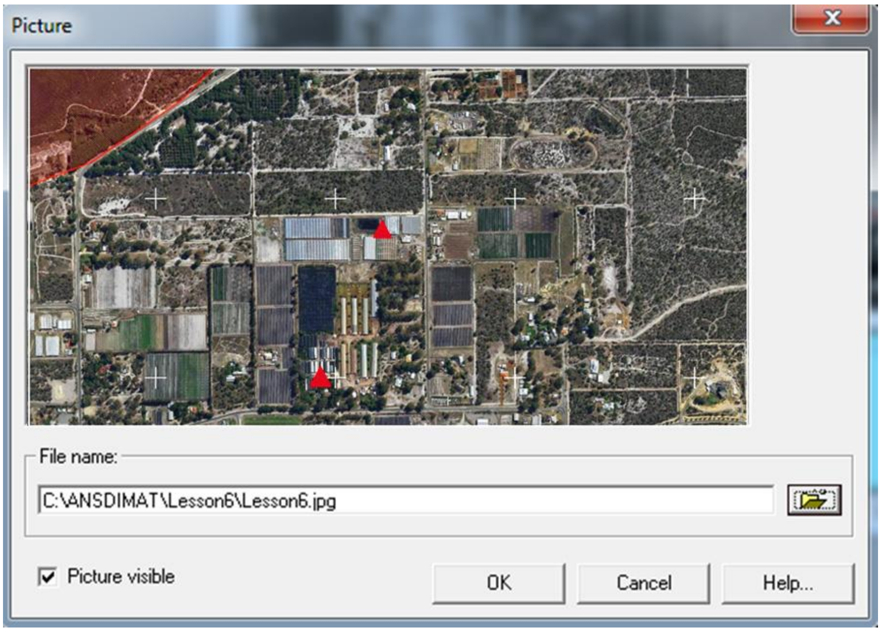



**Figure 5**. Picture selection dialogue window.

When importing the picture, check the box "*Picture visible*" to display it in the model map (Figure 6).

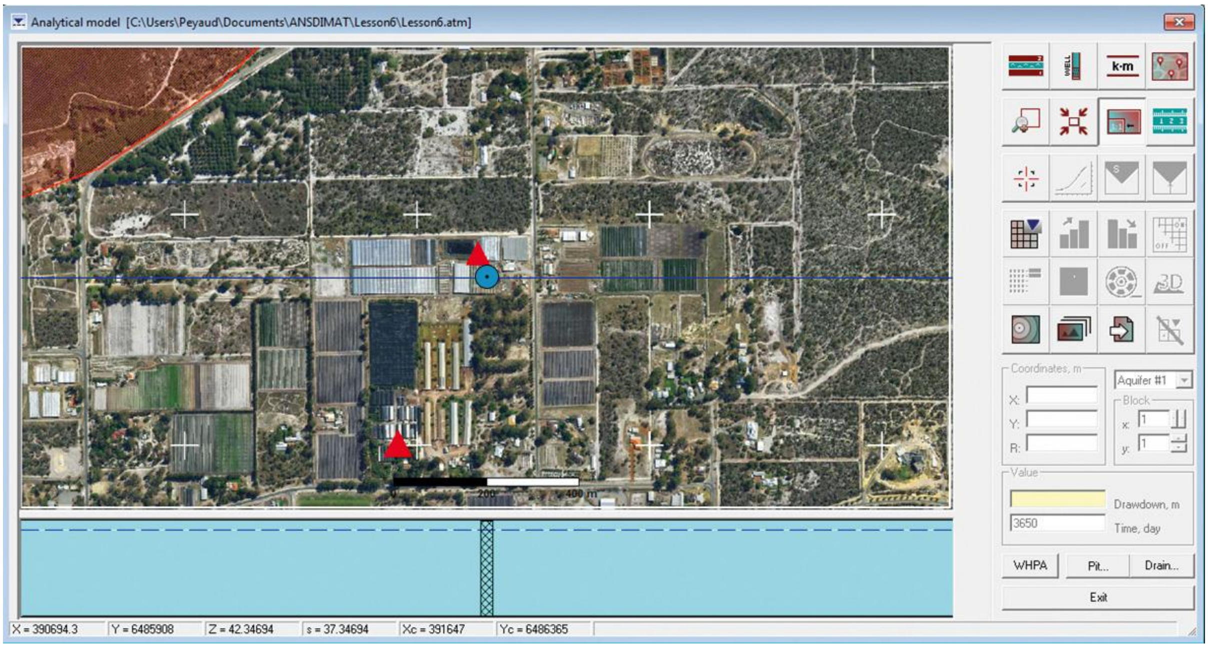
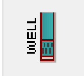



**Figure 6**. The analytical model window with the imported map.

### *Input wells in the model*

When a new AMWELLS project is created, it includes one well, located by default in the centre of the model area (Figure 6). This is because an AMWELLS project must include at least one well. In usual projects, this well would be relocated to one of the planned wells. For the purpose of the lesson, however, this well will be ignored while we create those we'll be using for the project. After that we'll remove the default well.

Wells can be added and relocated on the map using the "*Well*" icon  on the top row of the toolbar menu. After clicking on the icon, the "*Well and time*" dialog window opens (Figure 7) that allows editing existing pumping and observation wells, adding new wells and amending time intervals.

There are two methods to create a new well:

- Click on the "*Model*" button then left-click on the map at the location you want to create your well;
- Click on the "*New*" button and enter the well coordinates.

Let's use a different method for input each of the two wells.

1. Click on the "*Model*" button then left-click on the map at one of the proposed locations, let's say, the one in the north. You can adjust well location by clicking on the dot marking the well and holding while moving the dot. Call this well **P1**.
2. Press "*New*" button and rename the second well **P2**. Type the coordinates of the southern well in the dialog window (E: 391467; N: 6485998) – Figure 7.

Wells can be removed by a right click on the well symbol on the map or by using the "*Delete*" button in the "*Well and time*" dialog window. After you input two new wells, remove the default well **1w**. The window should look like Figure 7.

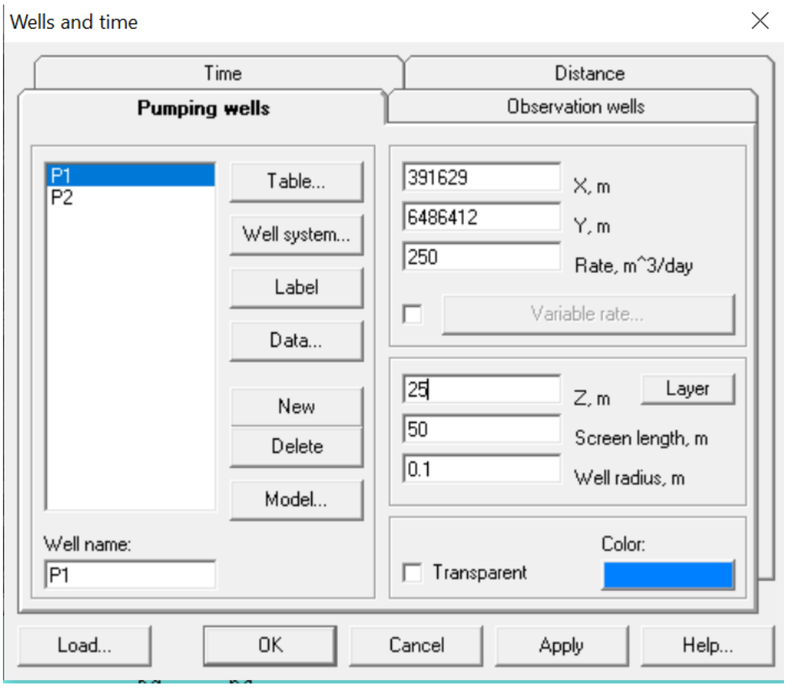



**Figure 7**. Well and time dialog window, pumping well tab.

In the "*Well and time*" dialog window, set the pumping rate for each well at 250 $m^3$/day and verify that the other parameters are correct:

- Z = 25 m
- Screen length = 50 m
- Well radius = 0.1 m

Check, by moving the cursor between wells, that each of the two wells has the same pumping rate, Z (distance between aquifer top and mid-screen), a well radius and a screen length. Alternatively, you can press the "*Layer*" button that will assign screen over the entire layer thickness. Your dialog window should look the same as that on Figure 7.

The "*Observation wells*" tab will not be used in this lesson. An observation well can be set-up in a similar manner to a pumping well. In the "*Distance*" tab, the distances between the wells are automatically calculated. There is no input required in this tab.

***Input aquifer properties***

Hydraulic conductivity and specific yield for this lesson are assumed respectively as 0.4 m/d and 0.1.

Click the icon 

to open "*Model parameters*" dialog window and enter the above values in "*Hydraulic properties*" tab (Figure 8). Aquifer is assumed isotropic, so vertical hydraulic conductivity equals to horizontal hydraulic conductivity (0.4 m/d). The storage coefficient and specific yield are equal to 0.001 and 0.1, respectively.

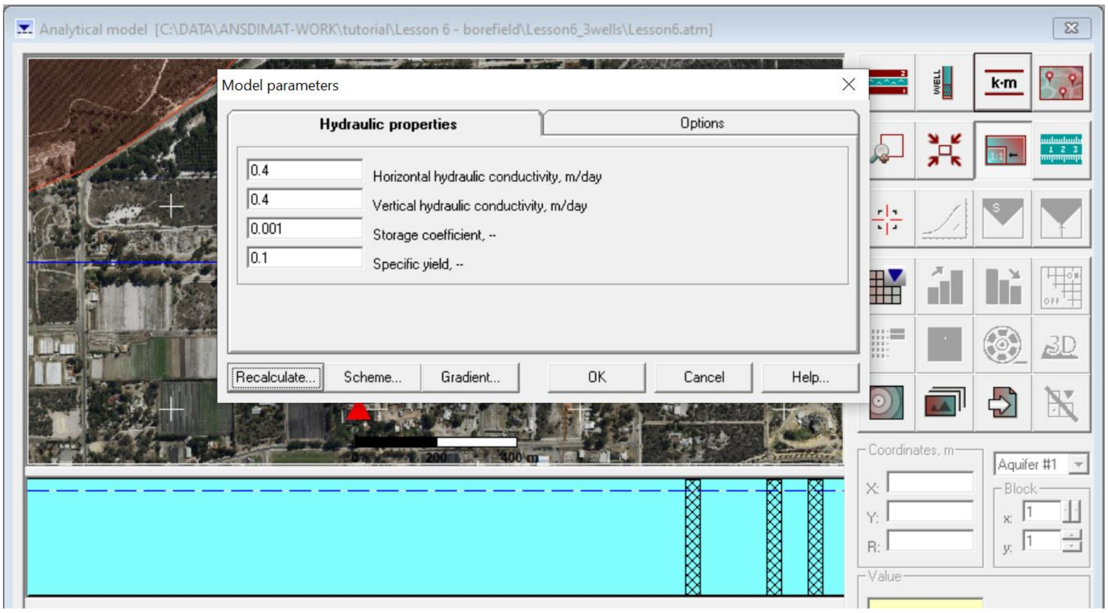



**Figure 8**. "Model parameters" dialog window.

**Define duration and time steps**

The last tab is the time input (Figure 9): this tab requires you to define simulation period and time steps. Input 3650 days (10 years) in the field "*Duration of pumping test*", as this will be the modelled period of the borefield operation.

Time steps can be entered by one of the four following methods:

- manually directly into the table;
- prepared in a spreadsheet and pasted in ANSDIMAT;
- automatically calculated based on equal time intervals (use field "*Number of time interval*" and button "*Calculate equal time intervals*";
- automatically calculated using the exponential formula (use field "*First time interval, day*" and button "*Calculate time measurements*".

For this Lesson, 11 steps for specific time intervals were created in an EXCEL spreadsheet and pasted in the Time tab (Figure 9), You can prepare the time series or copy it from the excel file that can be downloaded from the tutorial section.

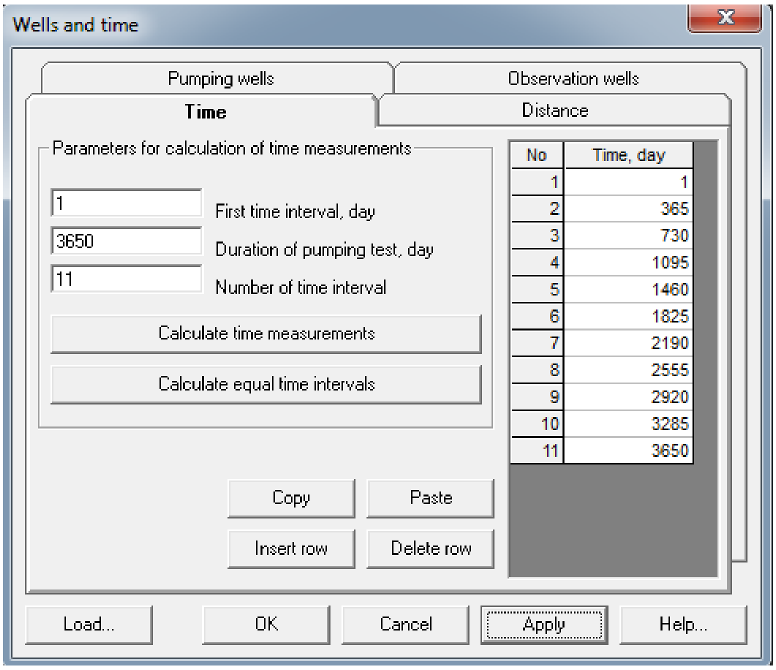



**Figure 9**. Well and time dialog window, time tab.

Once the wells and the time steps are entered, the model is ready to calculate. Figure 10 shows what the map should look like at this stage.

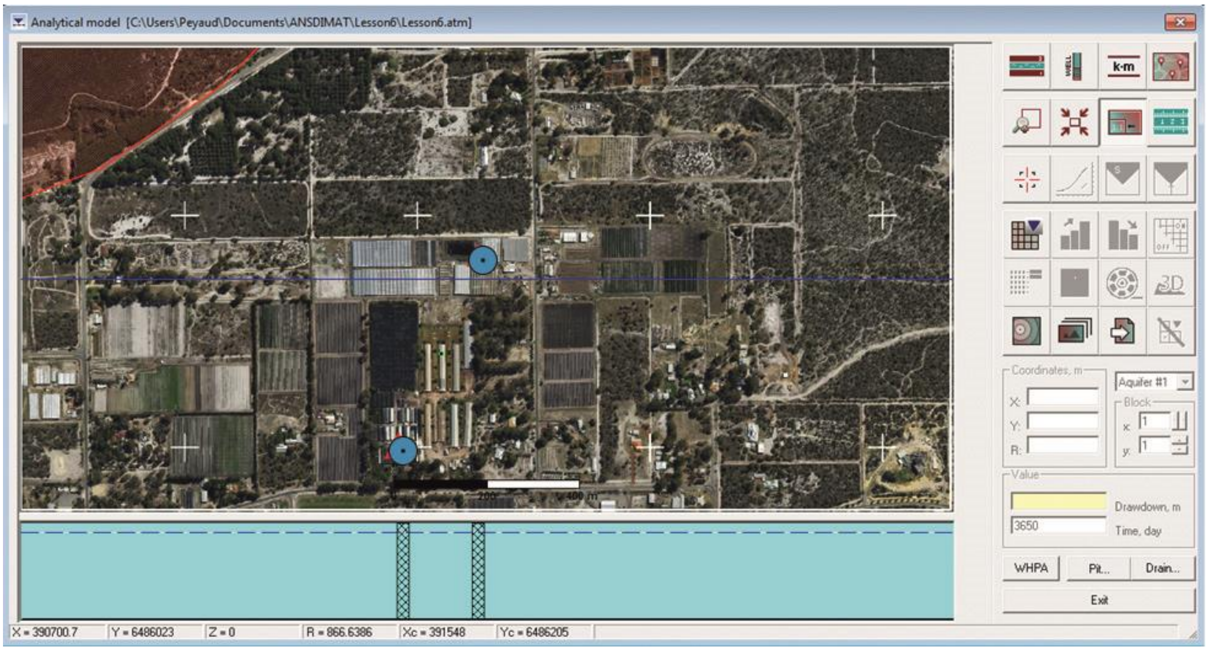
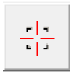
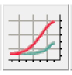



**Figure 10**. Graphical interface window showing location of the pumping wells (blue dots).

### *Calculating drawdown*

AMWELLS calculates drawdown for specified locations and time periods using selected analytical solutions (i.e. conceptual models).

Calculation results are displayed by a hydrographs (Figure 11 and Figure 12). To generate a hydrograph:

- Click on the "*Value*" icon  to enable the location selection; two lines appear on the map, their intersection marks the location, where the hydrograph is calculated;
- Click on the "*Plot*" icon  to generate the plot and view the results as time series.

Hydrographs can be generated for any location of the map. Check first at the locations of the pumping wells to verify that the well is not going to dry out during the production period. Figure 11 shows that actually both wells do get dry, as drawdown reaches 45 m (bottom of the aquifer).

Check whether drawdown complies to environmental requirements at the border of the protected area. Figure 12 presents an example of hydrograph. As you will be selecting a different location along the protected area border, your hydrograph may look slightly different.

Hydrographs are presented on a logarithmic time scale by default. This can be changed to a linear scale by unticking the "Log scale" box in the "Options" tab.

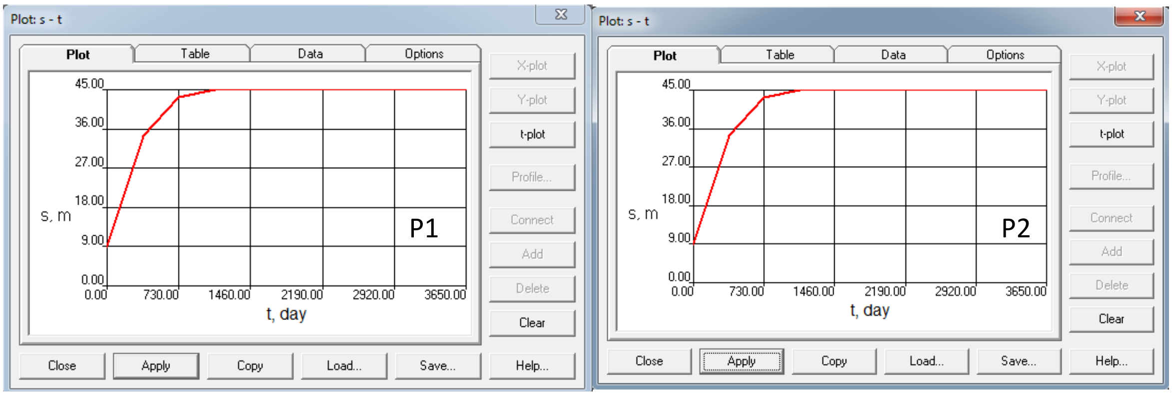



**Figure 11**. Drawdown vs time plots for the pumping wells.

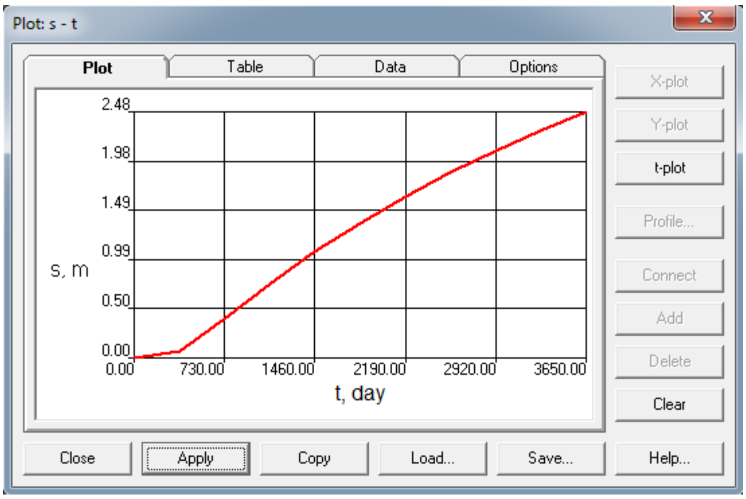



**Figure 12**. Drawdown vs time plot on the border of the protected area.

### Changing well locations

In this case, the hydrographs show a drawdown greater than 40 m in each well. This doesn't fulfill the required operational condition and, in fact, both wells would become dry. In addition, the drawdown at the border of the protected area exceeds the 1 m limit (environmental requirements breach).

To achieve acceptable drawdowns, you will have to move pumping wells further from the protected area and add one more borehole to decrease borehole individual pumping rates.

To move a borehole, left click on its marker to "grab" it and displace it to the location of your choice, then release the mouse button. Create one additional well. You can also move wells by changing their coordinates in X and Y fields of the Pumping wells dialog window. The proposed revised coordinates for this Lesson are:

- P1: E=392272, N= 6486090, rate=175 $m^3$/day
- P2: E=392572, N= 6486090, rate=175 $m^3$/day
- P3: E=392472, N= 6486090, rate=150 $m^3$/day

The revised borefield layout is presented on Figure 13. Each hole has the same depth, and radius, but it has a reduced pumping rate. Figure 14 and 15 illustrate that drawdown in each well and at the protected area border do not exceed their required limits of 40 m and 1 m respectively. This means that the revised borefield can supply 500 $m^3$/d during 10 year period under required conditions.

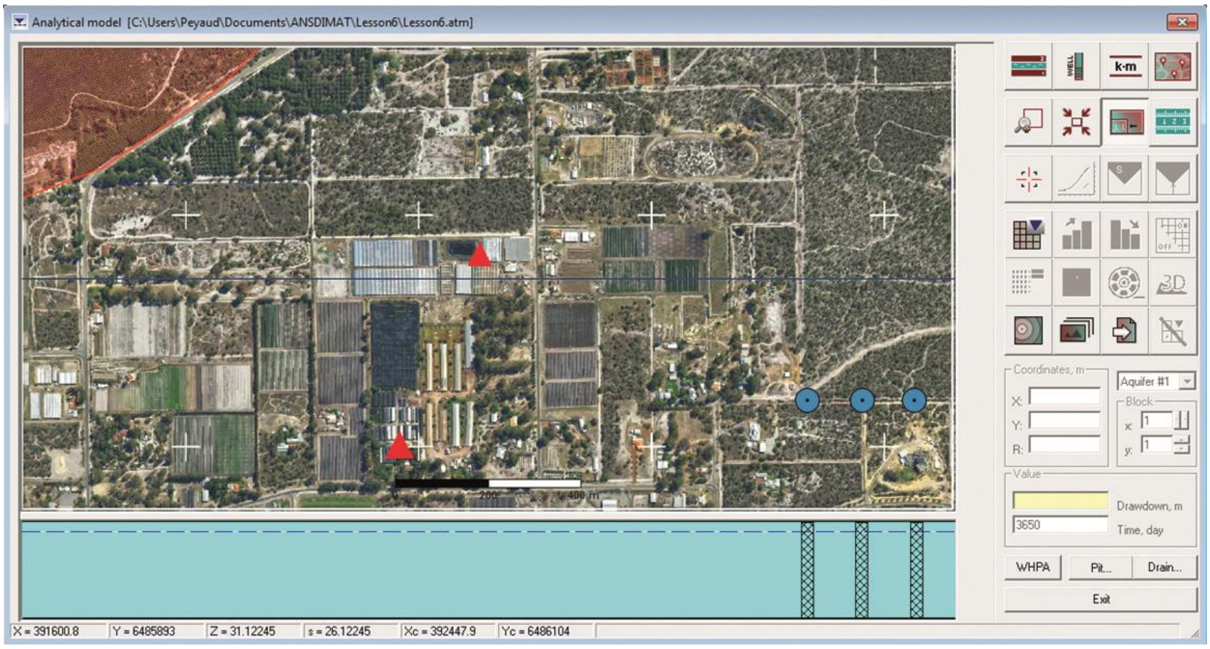



Figure 13. Revised proposal for the borefield.

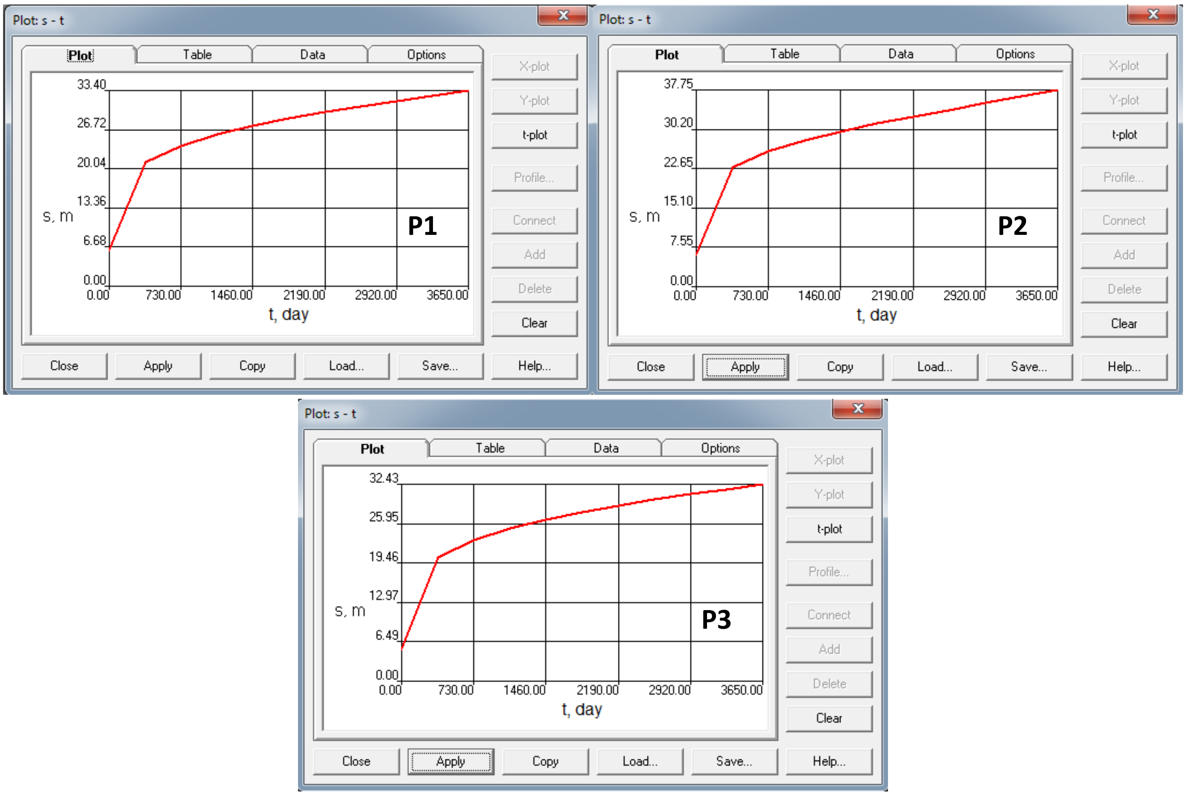



**Figure 14**. Drawdown vs time plot for the pumping wells.

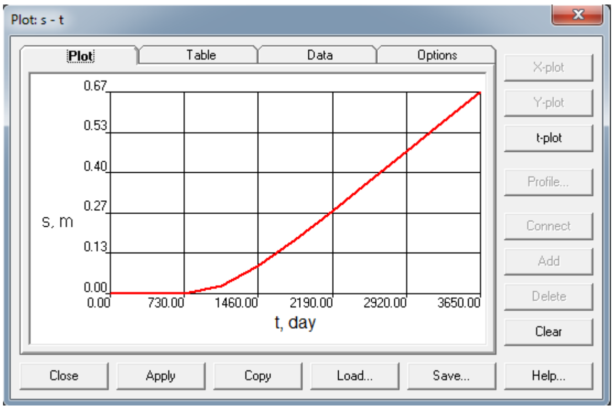



**Figure 15**. Drawdown vs time plot on the border of the protected area.

***Plotting contour maps and profiles***

AMWELLS can display drawdown contours on the map. The icon  opens the dialog window "*Groundwater piezometric map*" where drawdown or hydraulic head contours are plotted based on the range of calculated drawdown and the number of curves selected. The "*Contours*" tab (Figure 16 left) allows entering manually the number of contours and their values. It also allows selecting the colour of specific contours. Change the default value for the first contour to "1" so that your "*Contours*" tab look like Figure 16.

The "*Values*" tab (Figure 16 right) allows to calculate automatically the values at which contours will be drawn based on the number of curves selected, the maximum and the minimum contour value. Minimum and maximum values can be entered manually or AMWELLS will import these extremes from the model results. Tab "*Values*" also allows defining a colour scheme for the contours. The contour colour by default is a dark blue, in this case, it was changed to green and red. You can chose your own colour scheme or use the default one.

Click on the button "*Calculate contours*" to generate curves automatically. After that click the button "*Enter contours*" to apply them in the "*Contours*" tab. Values and colours can be overridden in the "*Contours*" tab, allowing to single out a specific contour for a specific value.

The third tab offers options for the interpolation and the drawing of curves. It is not used in this lesson.

After pressing the button "Apply", your plotted drawdowns should look like that on Figure 17. The map visually illustrates that drawdown will not exceed 1 m within the protected area.

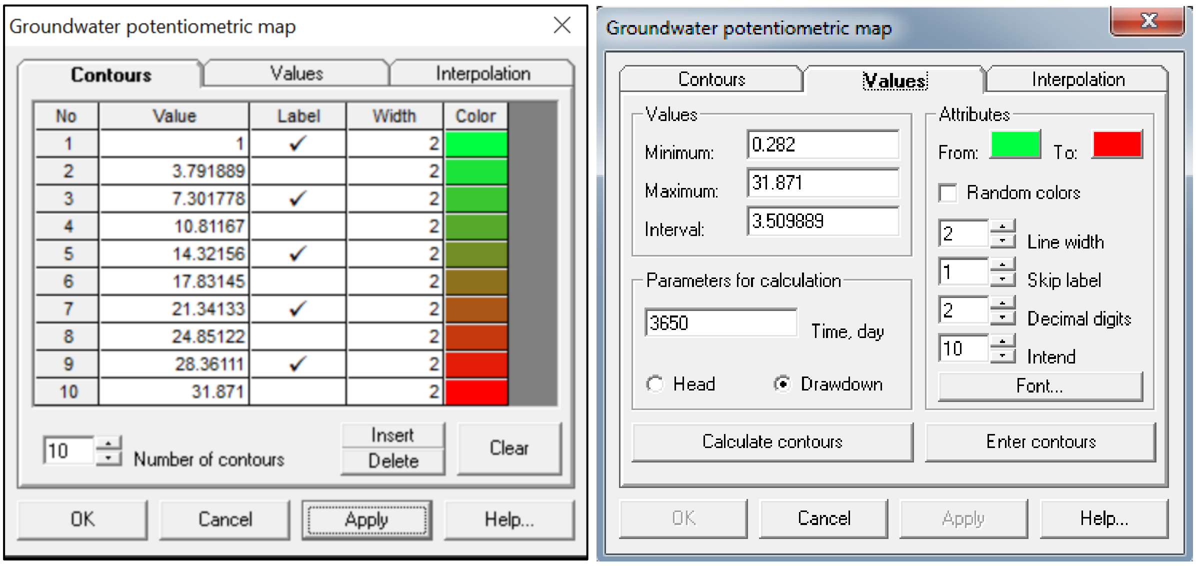



**Figure 16**. "Contours" tab with modified value of the first contour changed to "1" and "Values" tab with functions to automatize the generation of contours.

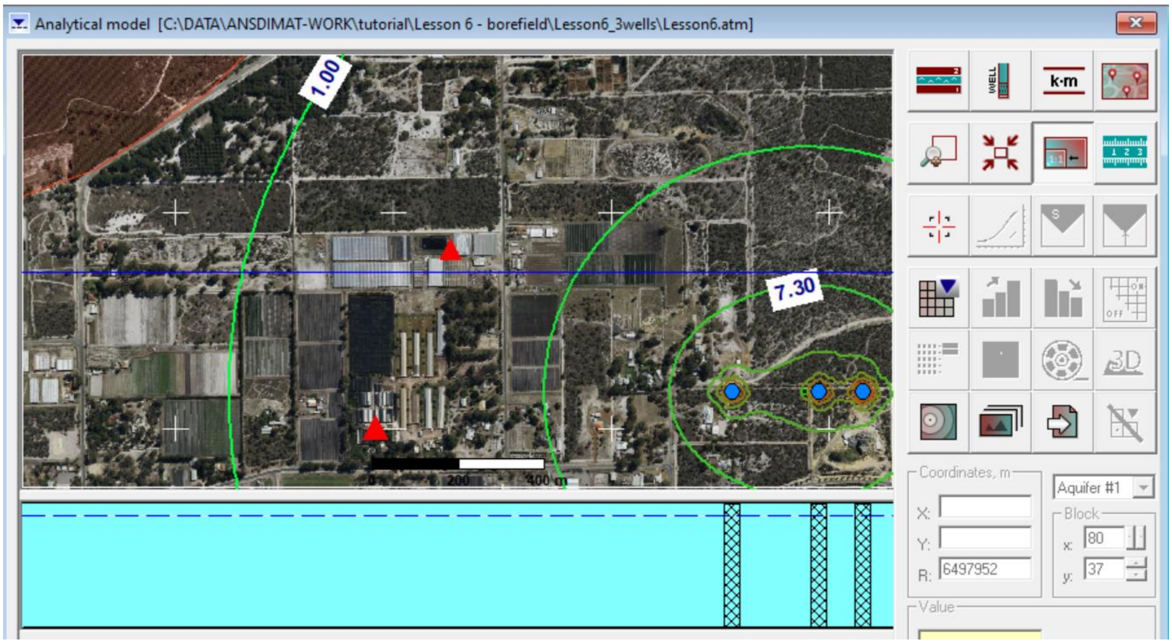
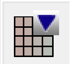



**Figure 17**. Drawdown map after 10 years of pumping from the revised wellfield.

AMSWELL can run the simulation on a grid to calculate drawdown/hydraulic head in each grid cell and produce a cross-section. The icon  of the toolbar menu opens the dialog window that creates a grid. AMWELLS then calculates the drawdown as a function of time for each cell to plot a drawdown or hydraulic head distribution map. The value for the cell is calculated at the centre of the cell.

Click on the icon  to open a window with two tabs: one displays the results of the previous run (if existing), the other tab enables to define/change the grid size (Figure 18). The "*Block size*" field displays the size of a cell based on the number of rows and columns. A larger number of rows and columns will result in a better map resolution, but will require a longer calculation time.

For this Lesson, let's increase the number of columns to 80 (field "*Number of blocks in X-direction*") and number of rows to 40 (field "*Number of blocks in Y-direction*"). Your grid size should be **25m x 25m**. When your Model Grid tab looks like Figure 18, press the button "*Run*". Once the calculation is finished, the drawdown map and water table profile (cross section) are displayed as presented on Figure 19.

You can change colours and classes of the map using Trace function (icon ).

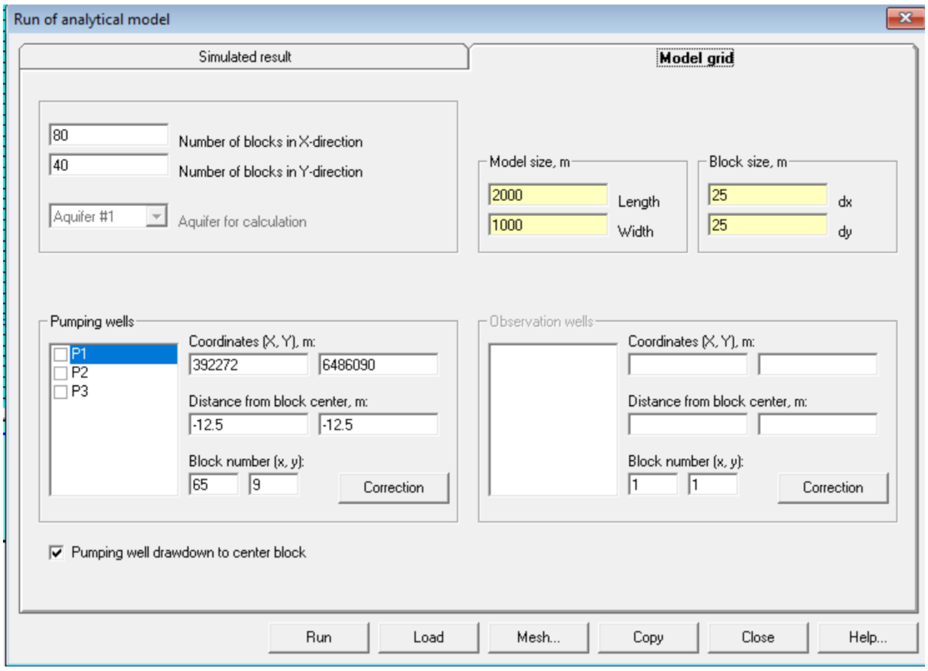



**Figure 18**. Model grid options for modified grid.

To draw a water level profile line through the borehole locations, enter a value of 9 in the field "*Block for cross section*" in the "*Model size*" dialog window (Figure 4).

The results can be exported as a table file (for instance a .csv file) and edited externally. To export results, simply click on the "*Copy*" button of the "*Run of analytical model*" window (Figure 18). The table that you can see in the "*Simulated result*" tab is copied to the Clipboard and can then be pasted in an excel file or any other table.

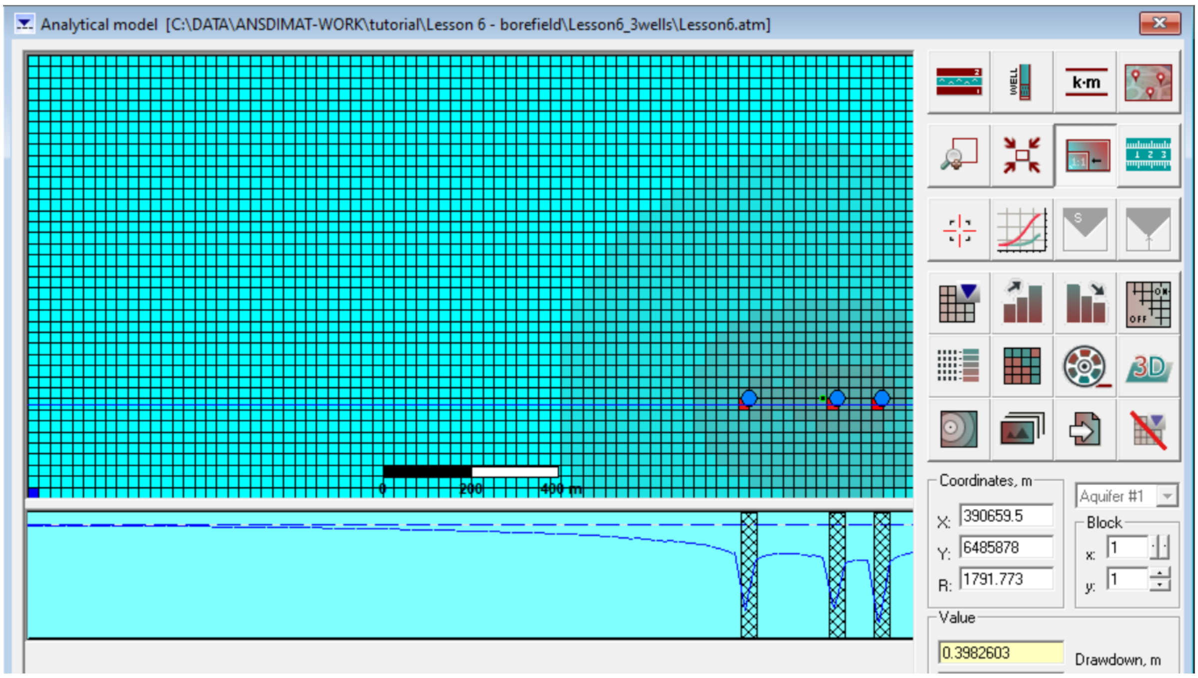
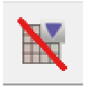



Figure 19. Analytical model calculated on the modified grid.

The icon  on the toolbar menu closes the grid mode and displays the map.

This concludes the Lesson 6 of the ANSDIMAT tutorial. The datafiles supporting this lesson can be downloaded from the ANSDIMAT web site where ready-made models are also available. Please note that for this lesson, two models are available: Lesson6_2wells corresponds to the initial design for the borefield, while Lesson6_3wells corresponds to the modified design.

If you have questions or feedback, feel free to email us at: support@ansdimat.com.