Flowpath II for Window 95/NT Demonstration Tutorial

Learning Objectives

- To examine some of the input and post-processing features and capabilities of Flowpath II for Windows™
- To run the numerical simulations for this problem
- To evaluate the output visualization features for the sample model results

Introduction

This demonstration tutorial guides you through some of the steps necessary to design and run, and then visualise the results of a model using Waterloo Hydrogeologic, Inc.'s Flowpath II for Windows™.

Problem Description

This example simulates a regional aquifer for predicting the impact of a solute spill on two municipal water supply wells. To evaluate the validity of the model results, you will compare the hydraulic heads calculated by the model, to observed groundwater levels measured in monitoring wells completed in the aquifer.

General Site and Hydrogeologic Setting

The study area encompasses an industrial subdivision, agricultural land and fish hatcheries. A river and several creeks provide surface water drainage.

The unconfined aquifer is comprised of Quaternary sands, and is underlain by relatively impermeable bedrock. Bedrock outcrops occur in the northwest and northeast corners and along the

southern edge of the study area. Water-level measurements show that groundwater flow in the aquifer is from the north to the east, but there is also a flow component from the west to the east.

Contamination Event

A chemical spill has occurred at a plant in the industrial subdivision and two municipal supply wells are potentially at risk due to the spill. The wells are located at a distance of 3000 ft and 6000 ft down gradient from the spill site.

Well Locations and Pumping Rates

Two municipal wells are pumping from the surficial aquifer in the study area. PW-1 is located at (x = 5330, y = 2200) and pumps at a rate of 300,000 cubic feet per day (ft^3/day) and PW-2 is located at (x = 8040, y = 3840) and pumps at 500,000 ft^3/day. During the tutorial, you will learn to refine the modeling grid around these critical locations to improve the finite difference simulation.

Hydraulic Conductivity Distribution

The hydraulic conductivity varies across the study area. Most of the area is underlain by a medium-grained, homogenous, and isotropic sand with a lateral hydraulic conductivity of 250 feet per day (ft/day). However, toward the southwest, the deposits grade into an area of finer-grained, fluvial deposits of silts and fine sands. Based on the results of pumping test analyses, these finer-grained deposits have hydraulic conductivities of 200 and 150 ft/day. The effective porosity across the site is assumed to be a uniform 25 per cent.

Aquifer Bottom Elevation

Based on the analysis of soil boring logs from the area, the aquifer bottom is located at an average elevation of 170 feet above mean sea level, and is horizontal across the study area,.

Groundwater Recharge

The average groundwater recharge is 10 inches/yr (0.0023 ft/d).

Terms and Notations

For this exercise, the following terms and notations will be used:

type - type in the given word or value
select - click the left mouse button where indicated
↵ - press the <Enter> key
↔ - press the <Tab> key to toggle between input boxes
☞ - click the left mouse button where indicated
☞☞ - double-click the left mouse button where indicated

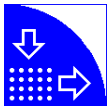
The **bold faced type** indicates menu or window items to click on or values to type in.

[….] denotes a button to click on, either in a window, or in the side or bottom menu bars.

File names to be entered are shown in the `Courier` font.

The icons to the left of instructions indicate that the icon performs the same actions as or is involved in the written instructions.

Starting Flowpath II for Windows™ 95/NT



To start Flowpath II for Windows™, simply double-click the Flowpath II program icon located on the Windows™ 95 Desktop. Now, you should be at the opening screen of Flowpath II for Windows™.

Model Input

Your first step in building a model is to create a new modeling project file.

Generating a New Model

To create a new model,

☞ **File**
☞ **New Model…**

The **Create New Project** dialogue box should be displayed with the insertion point in the Project Name field

In the Project Name box:

type: `Demo`

(Flowpath II for Windows™ will automatically assign a .PRJ extension onto the end of the filename, although this does not appear on the screen.)

Under Define model region

☞ ⊙ with base map

In the Map File section:

☞ **[Browse]**

A file selection dialogue box will appear prompting you to select a map file.

☞☞ **demo.bmp**[1]

In the Units section, choose the units which are appropriate for this project. In this case,

☞	⊙ **ft**	(Length**)**
☞	⊙ **ft/day**	(Conductivity)
☞	⊙ **ft^3/day**	(Pumping rate)
☞	⊙ **day**	(Time)
☞	⊙ **ft/day**	(Rech./Leakage)
☞	⊙ **ft^3/ft^2/day**	(Flux rate)
☞	**[Create]**	

The map file, demo.bmp will appear in the **Select Model Region** dialogue box.

Georeferencing .BMP (BITMAP) Base Map

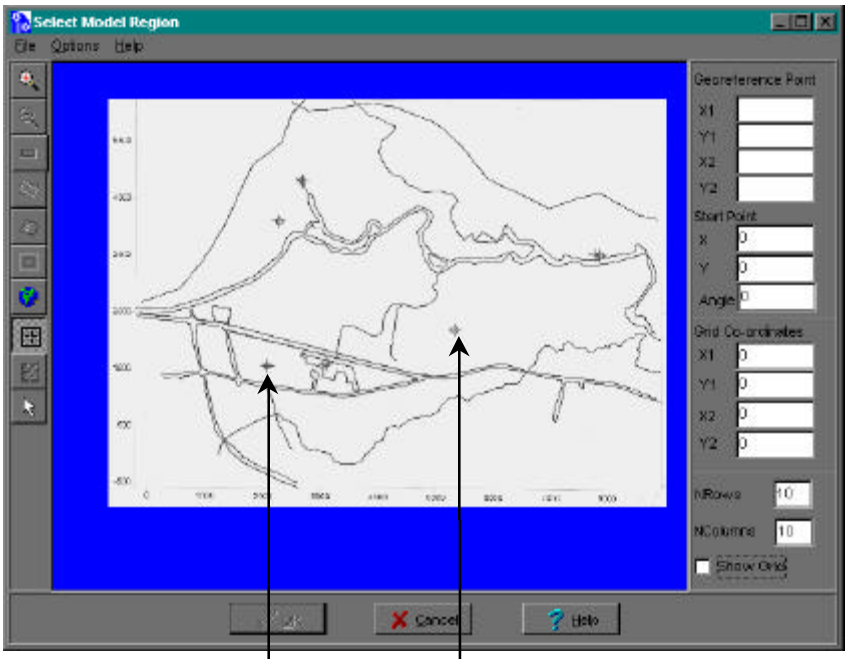
Many raster images, such as the Windows™ BITMAP, do not contain geographic control information. The base map which was scanned does not contain geographic control information and therefore must be georeferenced[2]. To georeference the map, you will use the X and Y co-ordinates of two wells within the study area with known co-ordinates. These georeference control points are shown in the figure below.

After you have georeferenced your BITMAP, and defined the project area, Flowpath II will create a new, georeferenced and resized .bmp map image for your modelling project. Although in this exercise you are using a black and white .bmp map image, Flowpath II for Windows™ allows you to use any .bmp file, e.g., scanned colour aerial

[1] Early distribution versions of Flowpath II for Windows™ did not include the Demo.bmp bitmap image file. If you do not have this file in your Flowpath II for Windows™ directory, choose the Demo.dxf file instead. If you are using the Demo.dxf file, after completing the **Units** section (see above), then proceed to the **Start Point and Grid Co-ordinate** section. Please contact WHI if you would like to download the Demo.bmp file from the company's FTP site.

[2] The establishment of a co-ordinate system with which the location of a point may be identified.

photographs and topographic maps, digital satellite and high altitude images.



Georeference Control Point 1 — Georeference Control Point 2

To assign the georeference co-ordinates to the bitmap, use the following procedure:

The  **Set Georeference Point** option is activated automatically whenever you are initially loading a raster image which does not contain georeference information.

Bring the mouse pointer into the Demo.bmp display area. Notice that the mouse pointer is transformed to a square with cross hair. This is the graphical **Set Georeference Point** tool. Carefully position this tool directly over the Georeference Control Point 1 (see above figure) and left-click. The X1, Y1 boxes turn red when you click to select the georeference point.

In the **Georeference Point** text box located in the upper right corner of the Select Model Region window, enter the following:

X1 Georef.	2017.34↔
Y1 Georef.	1555.45↔

To accept these co-ordinates for georeference Control Point 1,
↵ - press the <Enter> key

Notice that a red ‘star’ now appears over the control point.

Next, using the mouse again, position the georeferencing tool over georeference Control Point 2 and left-click.

In the **Georeference Point** text box located in the upper right corner of the Select Model Region window, enter the following:

X2 Georef.	5280.83 ↔
Y2 Georef.	2211.11↔

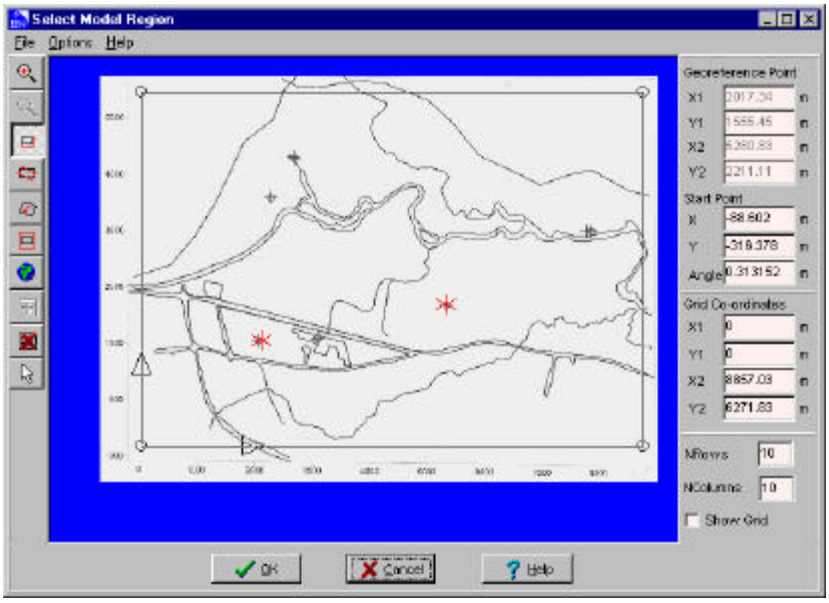
To accept these co-ordinates for georeference Control Point 2,
↵ - press the <Enter> key

You should now see a rectangle overlaying the Demo.bmp base map. This flexible map tool, known as the Select Model Domain Rectangle, allows you to select only the portion of the bitmap which is of interest for your particular project.

To define the model domain,

☞  **Resize Region Button** (from the left tool bar)

Using the mouse pointer, left-click and drag the corners of the **Select Model Domain Rectangle** until your base map image appears similar to the one shown below.



After you have defined the model domain, under **Start Point** and **Grid Co-ordinates** on the right of the **Select Model Region** dialogue box, check that the numbers listed correspond to the following list. If they are different, change them to match the following list.

X:	**-50**
Y:	**0**
Angle:	**0**

X1:	**0**
Y1:	**0**
X2:	**9000**
Y2:	**6000**

Next, specify the number of rows and columns for the finite difference model grid:

☞☞ **NRows** **50** ↔ (the lower right corner)
NColumns **60** ↔

To see the grid,

☞ ☑ **Show Grid** (in the lower right corner)

When you are finished,

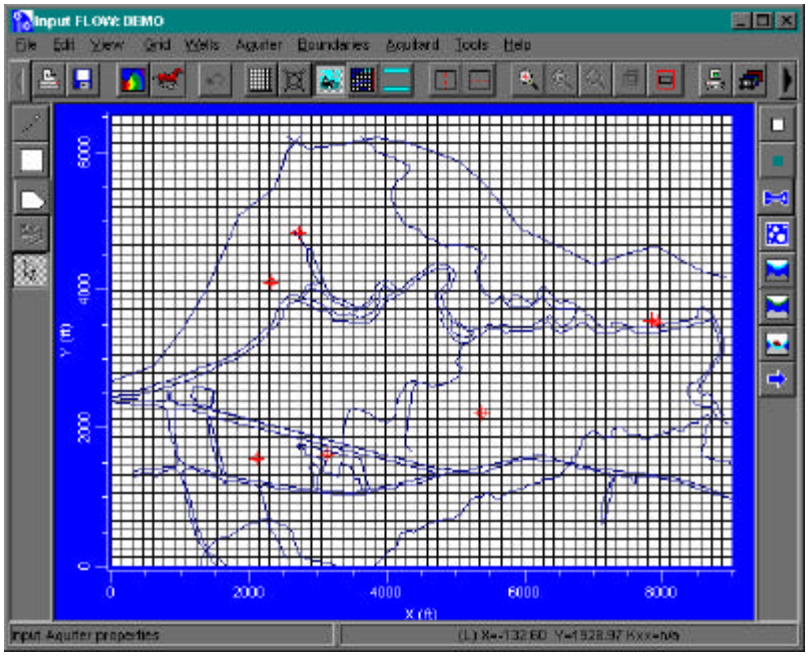
☞ **[OK]** (bottom of **Select Model Region** dialogue box)

In the **Save BMP** dialogue box which appears, enter the following image file name:

`Demonew`

☞ **[Save]** (to create a new georeferenced bitmap file)

Flowpath II for Windows™ will open the **Input** Module and display the new base map and model grid. The base map and model grid should appear similar to the ones shown in the following.



Adding Pumping Wells

Next, you need to add the two municipal wells to the model. To add pumping wells,



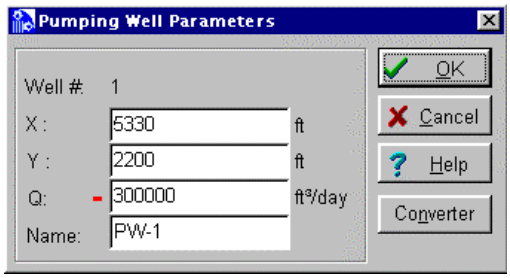
☞ **Wells** (from the Main menu)
☞ **Add Well,**
and choose **Pumping Well**



The **Save Data** dialogue box will appear.
Unless you choose otherwise, Flowpath II for Windows™ will prompt you
to save your work every time you change options or modules.

To make Flowpath II for Windows™ save all your work as you do it,
choose the following options under **Save Mode**,
☞ ⊙ **Save automatically**
☞ **[Save]**

With the left mouse button, click anywhere in the model area. A **Pumping Well Parameters** dialogue box will appear.



Double-click in the box to the right of **X** and enter the following:

X: **5330** ↔
Y: **2200** ↔
Q: **300000** ↔
Name: **PW-1**

☞ **[OK]**

Next, left-click anywhere in the model domain, and type the following information in the **Pumping Well Parameters** dialogue box which appears:

X: **8040** ↔
Y: **3840** ↔
Q: **500000** ↔
Name: **PW-2**
☞ **[OK]**

Refining the Grid

To better represent the areas around the wells, you should refine the grid in these areas. To do this, you will need to know the x-y co-ordinates to add grid lines. It is possible to display either the local co-ordinate or world co-ordinate location of your mouse pointer in the model domain.

☞ **Tools** (from the top menu bar)
☞ **Inspect**

In the **Inspect Properties** dialogue box which appears, under Show mouse co-ordinate,

☞ ⊙ **Local Co-ordinate System**

Under Inspect, make sure the Visible check box is checked.

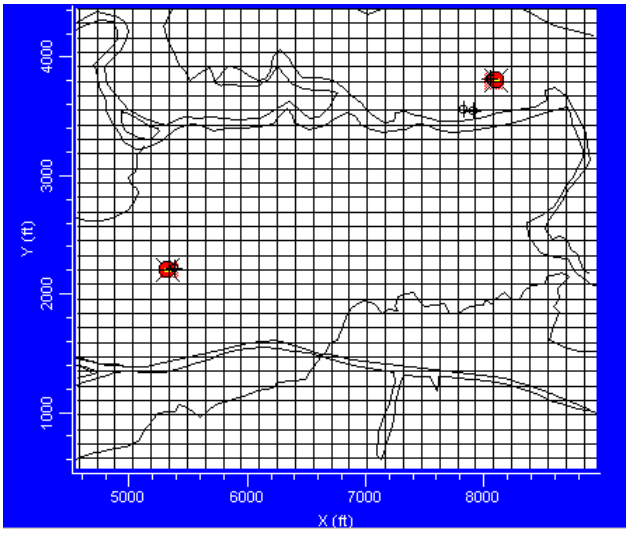
☞ **[OK]**

In the **Input Module** window, the status bar should now display the local co-ordinate location of your mouse pointer as it is moved in the model domain.

You need to zoom in so that you can refine the grid around the two pumping wells.

☞  (**Zoom-in** button from the top tool bar)

Now move the mouse pointer to approximately, X = 4500 and Y = 4500, and left click and drag open a window that encompasses the two wells. Release the mouse button. Your display should look like the following figure,





To refine the horizontal gridlines,

☞ **Grid** (from the top menu bar)

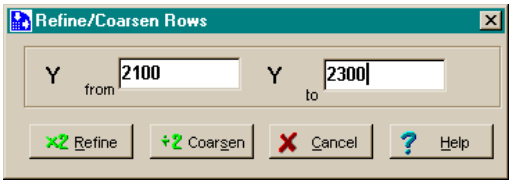
You are now in the Grid options of Flowpath II for Windows™.



☞ **Row**
☞ **Mark Rows**

1. We will mark grid lines and then refine them by a factor of two around the wells.
2. Left-click on a grid line 4 lines above the southernmost well and then on a grid line 4 lines below this well. A **Select Operation** dialogue box will appear,
   ☞ **[x2 Refine]**
3. Repeat these steps to refine the grid around the other well to the north.

4. Next, refine the grid immediately near the well. Right-click anywhere on the model domain.
   The **Refine/Coarsen Rows** dialogue box will appear.



   Type the following,

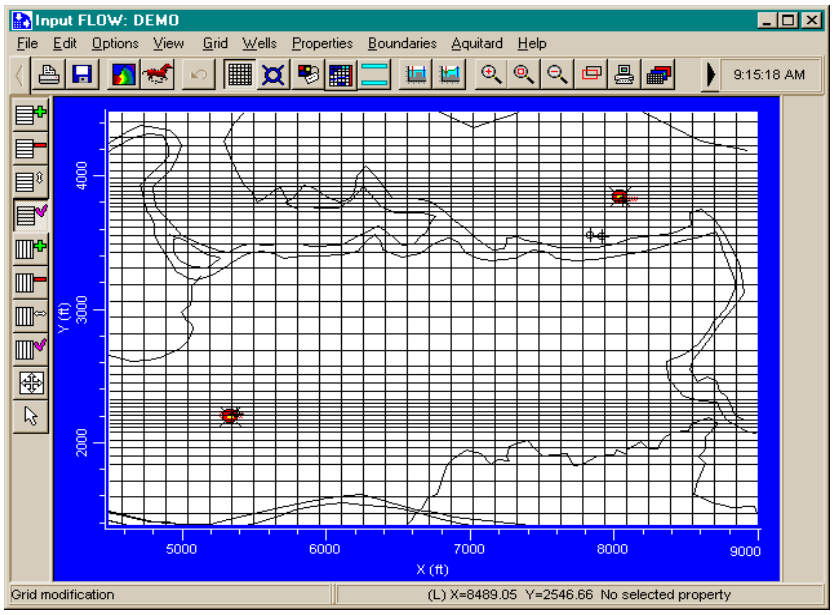
   Y_{from} = 2100 ↔
   Y_{to} = 2300
   ☞ **[x2 Refine]**

   Additional grid lines will be added between the co-ordinates specified.

   Now, refine the grid for the northerly well following the same procedures. Type the following,

   Y_{from} = 3750 ↔
   Y_{to} = 3950
   ☞ **[x2 Refine]**

Your model should now resemble the following figure.



Now refine the gridlines in the vertical direction.

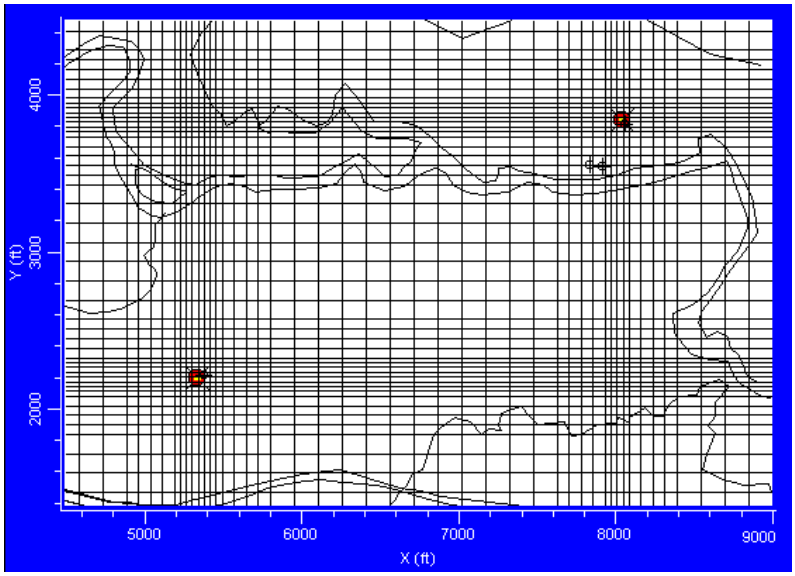


- ☞ **Grid** (from the top menu bar)
- ☞ **Column**
- ☞ **Mark Columns**

Use the same procedure as you used for the rows. First, mark the gridlines to right and left of both wells and refine by a factor of two. Then **right-click** anywhere in the model domain and refine further between the following co-ordinates.

X_{from}: 7950 X_{to}: 8150
X_{from}: 5200 X_{to}: 5500

The final grid should look like that below.





To see the whole model domain,

☞ **View** (**Zoom-out** from the top menu bar)
☞ **Zoom Out**

Input of Boundary Conditions

As previously stated, bedrock outcrops are located along the northwest, northeast and southern model boundaries. These boundary segments are assumed to be impermeable and will be designated as inactive regions, (i.e., no flow boundaries,) in the model. Areas of surface water and of groundwater flow which contribute water to the groundwater flow system will be designated as constant head boundaries.

Inactive Regions

You will now refine the model boundaries by eliminating areas that do not contribute to groundwater flow. You do this by assigning inactive cells. An inactive cell does not contribute or receive flow in the model.

To assign an inactive region:

☞  **Aquifer** (from the top menu bar)

☞  **Inactive Zone**

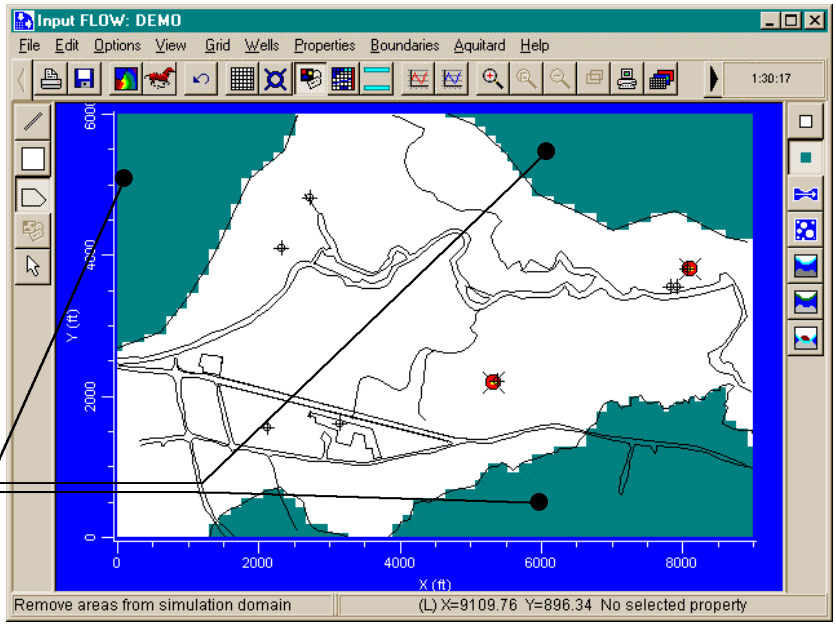
From the toolbar on the left side of the screen,

☞  **Draw Polygon** (draw polygon function)

This graphical tool will allow you to digitise polygons around inactive regions that do not contribute to groundwater flow. Move the mouse pointer to the top left corner of the model grid and left-click once to anchor the polygon. Move the mouse pointer down approximately 33 cells (Y=2700) and click again.

Getting Results Tips! Defining Inactive Regions

If the mouse pointer reverts to the arrow from a cross hair, your polygon will close prematurely. If this occurs, position the mouse pointer over the poorly defined inactive region, and right-click. From the context sensitive menu that appears, click **Erase**, then **[OK]** to remove the inactive region. Proceed with defining the inactive region as before.



Inactive Regions

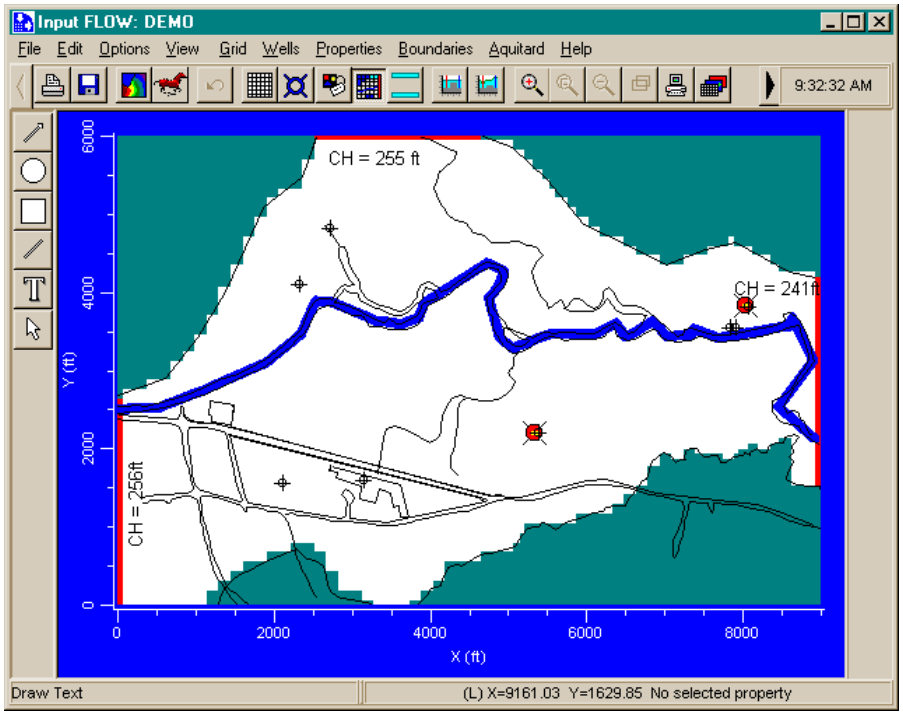
Continue this procedure until you have digitized inactive regions similar to those shaded regions shown in the figure above. Right-click to close the polygon. Repeat this procedure along the northwest, northeast, and southern boundaries using the diagram and the solid black lines on your background map as your guides.

You may want to turn the grid off so you can see the map more clearly. To do so,

☞ **View**
☞ **Grid**

Constant Head Boundaries

The next step is to assign constant head boundaries along the western, eastern, and northern sides of the model domain, as shown in the figure below.





☞ **Boundaries** (from the top menu bar)

A drop-down menu will appear listing all of the available Flowpath II for Windows™ boundary conditions that you can assign to your model.



☞ **Constant Head**

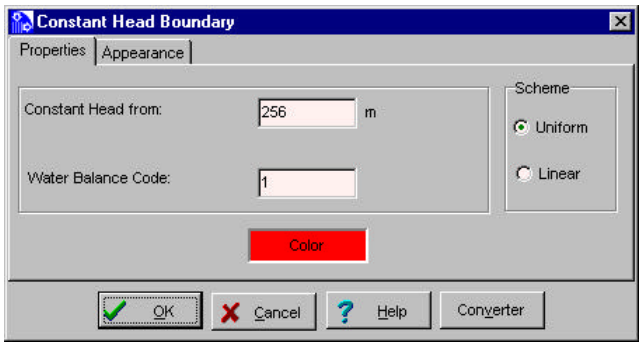
The **Constant Head** toolbar will be displayed on the left of the window that you will use to graphically assign constant head boundary conditions as lines, polygons or windows. To assign a constant head boundary along the western boundary of the model domain, using the Draw Line tool,

☞  **Assign Line** (from the left tool bar)

Move the mouse pointer to the top of the active western boundary (x = 0, y = 2650) and click once to anchor the line.

Then move the mouse pointer to the bottom left corner and right-click to close the line. The cells corresponding to the line will be shaded pink, indicating they will be assigned a constant head boundary condition.

A **Constant Head Boundary** dialogue box will appear prompting you to enter the required constant head information.



The water balance code is used to keep track of assigned boundary conditions for water balance calculations and is automatically set to one.

Constant Head from: 256 ↔
Water Balance Code: 1

☞ **[OK]**

The constant head line should now be red.

Now move the mouse pointer to the eastern boundary. Click at the top of the boundary (y = 4200) to anchor the line. Move the mouse pointer to the bottom right corner and right-click to close the line. In the **Constant Head** dialogue box,

Constant Head from: 241 ↔
Water Balance Code: 2

☞ **[OK]**

When you finish inputting the constant head on the eastern boundary, move to the northern boundary. At the northern boundary assign a constant head of **255** feet and a water balance code of **3**.

River Boundaries

We will now simulate the river that runs from west to east across the site from a surface water elevation of 255 feet in the west to 235 feet in the east.



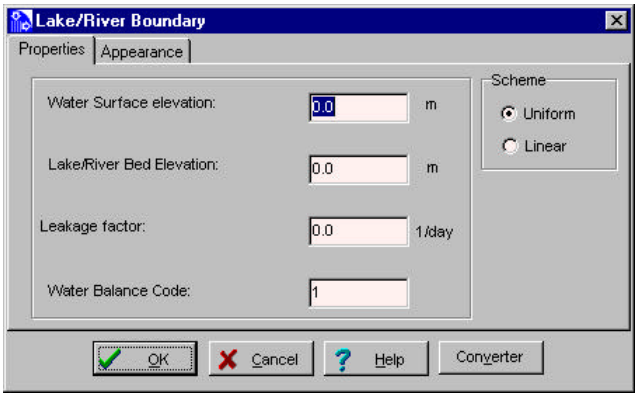
☞ **Boundaries** (from the top menu bar)
☞ **Lake/River**

From the side toolbar, choose the Draw Line option,

☞  **Assign Line** (from the side button bar)

You will now digitise a line to represent the river. Left-click at x = 0, y = 2500. Following the outline of the river on your .bmp base map, digitise the length of the main river channel. Left-click at every change in the river's course, and at the end of the river (x = 9000, y = 2100), right-click to close the line.

The **Lake/River Boundary** dialogue box will appear prompting you to enter the Lake/River information.



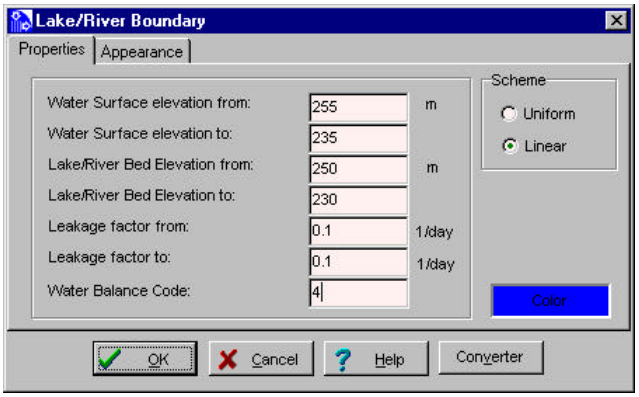
By default, Flowpath II for Windows™ assumes each surface water node will be identical (uniform scheme). For this model, however, the values of the nodes will change as you move along the constant head node.

In the physical system, this means the water level elevation (head) along the length of the river is not constant, and varies due to a change in surface elevation along the river's course. Flowpath II for Windows™ uses linear interpolation along the river's course to determine the variable head values using the beginning and ending head elevation values you specify.

☞ ⊙ **Linear**

Double-click in the box to the right of **Water Surface from:** and enter the following information:

Water Surface from:	**255**
Water Surface to:	**235**
Elevation of the Lake/River bed from:	**250**
Elevation of the Lake/River bed to:	**230**
Leakage factor from:	**0.1**
Leakage factor to:	**0.1**
Water Balance Code:	**4**



Under the **Appearance** tab,

☞ ⊙ **Contour**

Increase the Thickness to **7**

☞ **[OK]**

You should now turn the display of the model grid back on. This time use the **Overlay Control** which allows you to turn various element overlays on and off at once or to rearrange the overlay order.



☞ **View** (from the top menu bar)

☞ **Overlay Control**
☞☞ **Grid**

If you click on the **Order** tab, you can change the order in which the overlays are displayed.

☞ **[OK]**

Now the grid should be visible. Next, you assign the aquifer hydraulic conductivities and porosity.

Input of Aquifer Properties

From the top menu,

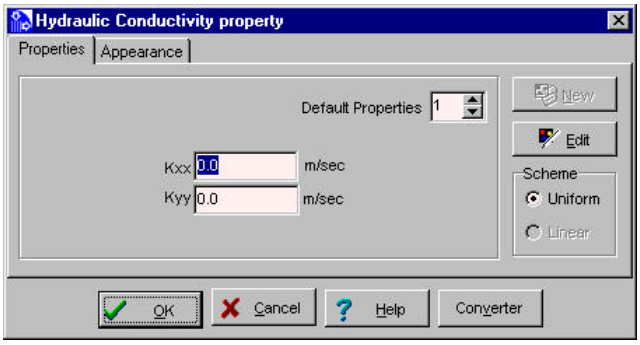


☞ **Aquifer**



☞ **Conductivity**

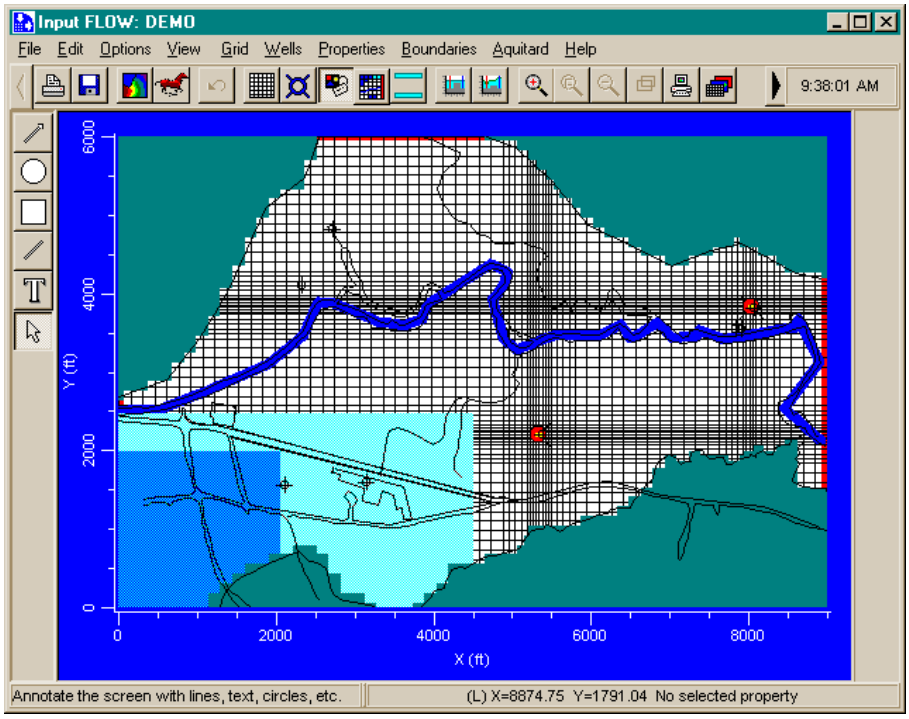
By default, the hydraulic conductivity of the entire model (Property 1) is set to 0.



Double-click in the box to the right of **Kxx** and enter the following,

Kxx **250** ↔
Kyy **250** ↔
☞ **[OK]**

In the southwestern (lower left) corner of the study area, the unconsolidated deposits are less permeable, with Kx = Ky between 200 ft/day and 150 ft/day. You will now redefine the conductivity values in these regions, use the following figure as a guide.



☞  **Draw Rectangle** (from the left toolbar)

Move the mouse pointer to around x = 4500, y = 2500, click and drag a rectangular area while moving the mouse pointer to the model origin (0,0); and release the mouse button. In the **Hydraulic Conductivity property** dialogue box that appears,

Kxx **200** ↔
Kyy **200** ↔
☞ **[OK]**

Now define the second low-permeability area. Left-click at about x = 2000, y = 2000 in the model domain to anchor the selected area. Then release the mouse button at the origin (0,0) to close the window.

In the **Hydraulic Conductivity property** dialogue box that appears,

☞ **[New]**
Kxx **150** ↔
Kyy **150** ↔
☞ **[OK]**

To enter the aquifer porosity



☞ **Aquifer** (from the top menu bar)
☞ **Porosity**

In the dialogue box that appears,

Porosity: 0.25
☞ **[OK]**

To enter the elevation of the bottom of the aquifer,



☞ **Aquifer**, (from the top menu bar)
choose Elevation
☞ **Bottom**

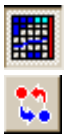
In the **Elevation Bottom Property** dialogue box that appears, type the following value

Elevation Bot. 170
☞ **[OK]**

Since the aquifer is unconfined it is not necessary to input the elevation of the top of the aquifer.

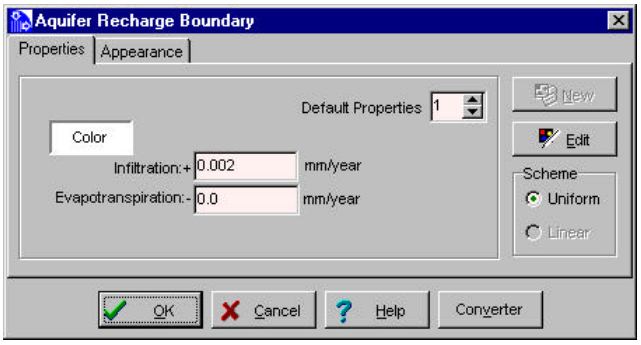
Input of Net Groundwater Recharge

From the top menu,



☞ **Boundaries**

☞ **Recharge**



Assume that the net groundwater recharge (infiltration minus evapotranspiration) is uniform for the entire aquifer at a value of 10 inches/year (2E-3 ft/d). Enter the following in the **Aquifer Recharge boundary** dialogue box:

Infiltration (In:+): **2E-3**
Evaporation (Ev:-): **0.0**
☞ **[OK]**

You have completed inputting all of the aquifer boundary and flow properties.

Input of Observation Wells for Calibration

Every groundwater flow model must be calibrated to ensure that it is a reasonable representation of the actual flow conditions in the aquifer. One of the most common means of calibration is to compare the water levels measured at observation wells in the field to the calculated water levels for those points in the model. In this way, you can assess the validity of your modelling results and establish the model's credibility as a predictive tool.

Adding an observation well is very similar to adding a pumping well, as in Section 2.



☞ **Wells,** (from the top menu bar) choose **Add Well**, then



☞ **Observation Well**

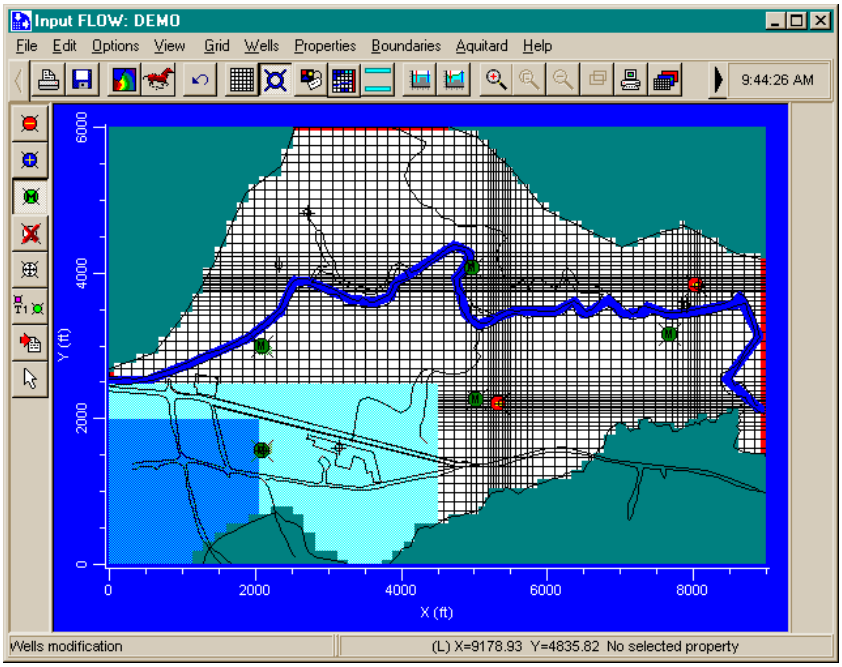
Left-click anywhere on the model domain and **the Observation Well Parameters** dialogue box will appear. Type the following,

Well #:	**1**
Xw:	**2089** ↔
Yw:	**3000** ↔
Hw:	**250.0** ↔
Name:	**OW-1**
☞	**[OK]**

Repeat this procedure four more times to add the remaining 4 wells based on the data in the following table

Name	Xco-ordinate Xw	Yco-ordinate Yw	Observed Head Hw
OW-1	2089	3000	250.0
OW-2	2089	1554	250.1
OW-3	5035	2250	237.2
OW-4	7669	3161	235.2
OW-5	4982	4071	244.8

The following figure shows you where your wells should now be located.



Running the Model

THIS DEMONSTRATION PROGRAM DOES NOT ALLOW RUNNING OF THE NUMERIC SIMULATION.

Having constructed your model domain, specified aquifer flow properties and constant head boundaries, and placed pumping and observation wells in your model, you are now ready to setup and run the numerical solver



- ☞ **File** (from the top menu bar)
- ☞ **Run**

The **Change options, Run solvers** dialogue box should be displayed. Switch to the **Flow** Tab and select the following options,

- ☞ ⊙ **PCG**
- ☞ ⊙ **unconfined**
- ☞ ⊙ **non-leaky**

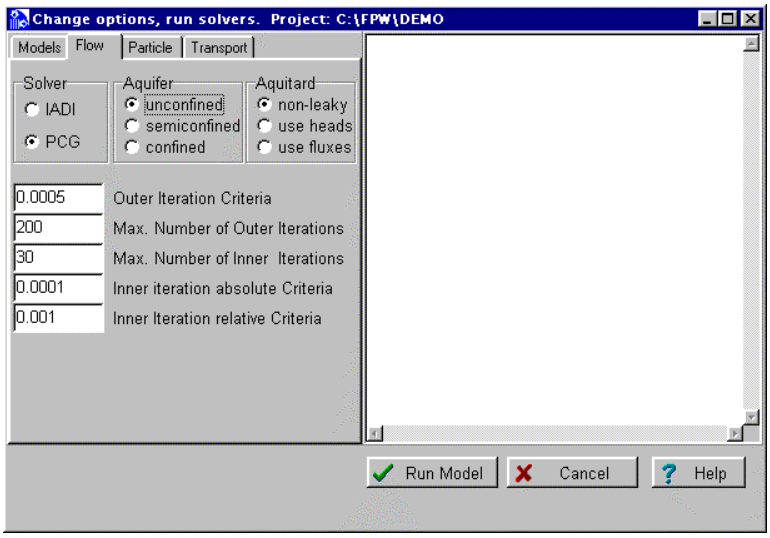
The default convergence parameters for the numerical solver will be sufficient for this simulation.

Next, on the **Models** tab, under **Choose Models to Run**, click to select the **Flow Model**, and **Particle Tracking** options.

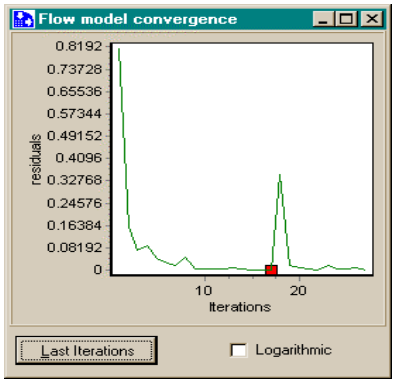


To run the model,

☞ **[Run Model]** (from the bottom of the dialogue box)



As the model runs, the iterations and residuals are shown in the right window and a plot of the residuals (shown below) is shown on the left.



When the solver is finished, the iterations stop and you can look at the report of the iterations and their residuals by using the scroll bar. To reach the Output Module,

☞ **[Close]**
☞ **[Yes]** (to save the options for the next run)

You are now in the **Output Module**.

Output Visualisation

The graphical output features of Flowpath II for Windows™ have been carefully designed to allow you to visually analyse your modelling results. You can select just the right combination of output features to create project-specific maps and graphs.

Equipotentials and Contouring Options

By default, the head equipotential contours are displayed upon entering the Output Module. To modify the display of head equipotentials, you must first activate the Head options layer.

From the **Output Module** toolbar (top of window)

☞ 

The Equipotential and Contouring layer is now active. A tool bar list is now displayed on the left of the window, which will be used to add and delete contour lines, as well as, move or erase contour labels.

☞  (from the left toolbar)

Now left-click where you would like to insert a contour line.
To move a contour label,

☞  (from the left toolbar)

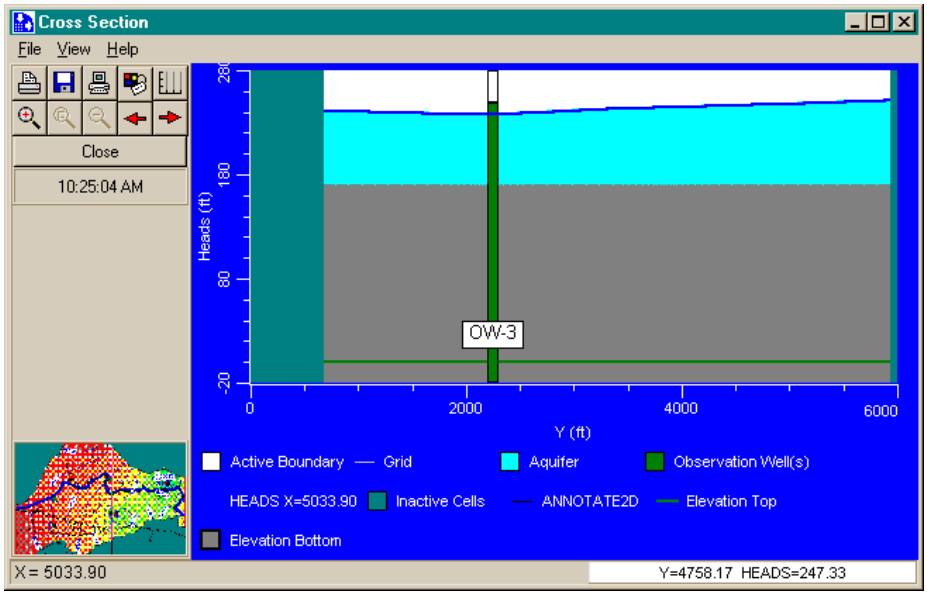
Click any contour and the closest label will be moved to the current position of your mouse.

Flowpath II for Windows™ can generate a cross-section of the head equipotentials. To see a cross-section of your head values,



☞ **View**
☞ **X-section**

Move your mouse pointer into the model domain and a purple line will appear showing where the cross-section will be drawn. Left-click anywhere in the model domain and the Cross-section window will appear (shown below).



You can change the Graphic Settings, such as colours, location of the cross-section, or the line style, by choosing



- ☞ **View**
- ☞ **Property**

Change the settings as you wish and when you are finished

- ☞ **[Close]** (to exit the Graphic Settings screen)
- ☞ **[Close]** (to exit the cross-section mode)

Viewing Flow Velocities

To confirm the trends you have interpreted from the heads contours and colour shading, you can display the flow velocities. The flow velocities can be viewed either as vectors (showing magnitude and direction) or as a colour shading map. To activate these options,



☞ **Options** (in the Output Module menu)
☞ **Velocities**

On the **Velocity Settings** dialogue box which appears, on **the Vector Control** tab, click the **Vectors Visible** box to display velocity vectors on your model. Change the **# vectors per Grid direction** so that they do not overlap.

Type: 40

The vectors are plotted all in one colour, which can be changed by clicking **[Vector colour]** and choosing an appropriate colour. There is also a colour shading option, which shades the vectors based on their magnitude.

☞ **[OK]**

To better view the velocity vectors, turn off the head contouring overlay:

☞ **View**
☞ **Overlay Control**

Under the Overlay tab,

☞☞ **Outputs** folder
☞☞ **Head Contours** **(to clear the check box)**
☞ **[OK]**

The model domain can be colour shaded the same way it is done for the display of heads or drawdown.

Particle Pathlines

In Flowpath II for Windows™, you can place particles anywhere in your model and then animate their movements throughout the model under the influence of the simulated flow conditions. These particles can be deleted and reinserted without having to recalculate the model.

To activate the Particle Pathline option,



☞ **Options** (in the top menu bar)
☞ **Particles**

NOTE: All of the particle functions and options can only be selected from the left toolbar.

In the **Pathlines options** dialogue box which appears, on the **Particle Defaults** tab, under **Wells**,

☞ ⊙ **Reverse**

Under Number of Particles,

Type: 6

On the **Calculation/Animation** tab,

☞ **Backward Tracking** (select this option)
☞ **[OK]** (from the bottom)

To activate the Particle toolbar,

☞  (**Particles)** (from the top tool bar)

Insert a reverse set of particles around a pumping well,

☞  (**Draw Circle)** (from the left toolbar)

Then click close to any pumping well and drag the mouse pointer to define a small radius of particles around the well. Make the radius of the particle circle as small as possible around the wellhead.

☞ **[OK]** (in the Well Particles Properties dialogue box)

The pathlines must now be calculated .

☞  **(Calculate Particle Pathlines)**

Next, to activate the Animation option,

☞  **(Animate Particles)** (from the left toolbar)

The **Particle Video Player** will appear. Click  (play) button to see the particles move.

You can insert lines of particles.

☞  **(Draw line of Particles)**

Place the lines of particles along the north and west boundaries where you inserted the constant head lines. Click the bottom left corner of the model domain and drag to the top of the active cells and click again. Do the same for the northern constant head boundary.

Then click the  **Calculate** button to have Flowpath II for Windows™ calculate the pathlines of the new particles you added. Click the  **Animation** button to prepare the particle and pathline simulation.

Click the  Properties button from the left toolbar, and under the **Calculation/Animation** tab ensure that the **Backward Tracking** option is selected. On the Particle Defaults tab, under Lines, click to activate the Reverse scheme. Click the  **Play** button to play the new animation.

You can delete individual particles or all of the particles using the following buttons from the left toolbar:

 and  respectively.

Close the Pathlines Video Player by clicking the **Close** button.

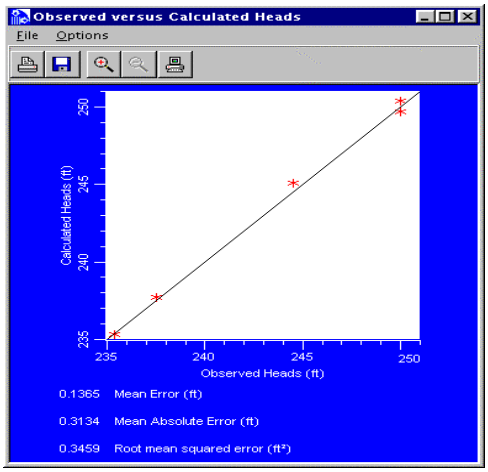
Viewing the Calibration Graph

In the Input section, you added 5 observation wells to assist you in calibration in the **Output Module**. An Observed versus Calculated heads graph can be plotted by,



- ☞ **Calibrate**
- ☞ **Calc. Vs Obs.**

The Observed versus Calculated Heads dialogue box should appear.



You can zoom in or out to change the plotted scale. If you click any data point a well property window will appear. You can change this so that when your mouse pointer passes over a data point the well information appears by,

- ☞ **Options**
- ☞ **Pop-up well info**

Try out this function.

To close the Calc. vs. Obs. graph,

- ☞ **File**
- ☞ **Exit**

You can also output this graphic to a .bmp file, the clipboard or to a printer, in the **Print to file** option.

☞ **File**

Under Save as Type, click the down arrow to view the file types available to save your model output.

☞ **[Cancel]** (to close the Save as dialogue box)

These functions are the same as those found in the **Input Module** under File.

Annotating your Graphic and Printing

Flowpath II for Windows™ will produce report-quality maps and graphs either to a printer, the clipboard, or save your graphic in a .DXF, .BMP or .WMF file format. Saving your output in a different graphic file format gives you the flexibility of importing it into other graphic programs for additional post-processing. Before you printout the graphic, you can annotate it with shapes and text which can help you to describe the graphic to your targeted audience and draw attention to keys areas of the model or graph. To add text to your map,

☞ **Edit**
☞ **Annotate**
☞ **Add Text**

Click close to the river and the **Text Properties** window will appear.

Type: Souhegen River (in the **Text** field)

Unless you would like to change the font or the angle of the text (0 is horizontal),

☞ **[OK]**

Other functions include drawing circles, rectangles and lines. All of the above can be any colour you choose and by clicking the right mouse button you can edit their size or erase them. This ends the Flowpath II for Windows™ tutorial.

To exit Flowpath,

- ☞ **File**
- ☞ **Close**
- ☞ **[Yes]** (to save the settings)

In the Main Screen,

- ☞ **File**
- ☞ **Exit**
- ☞ **[Yes]** (to close the input screen)