

DR. SHUGUANG LI AND ASSOCIATES

INTERACTIVE GROUNDWATER MODELING 3 (IGW 3) QUICK TUTORIAL

This tutorial is designed to quickly acquaint the first time user with the functioning of the software and the steps involved to create and solve a model.

Figure 1 shows a sample of the IGW 3 Working Area and acts a reference for the steps to perform in this quick tutorial.

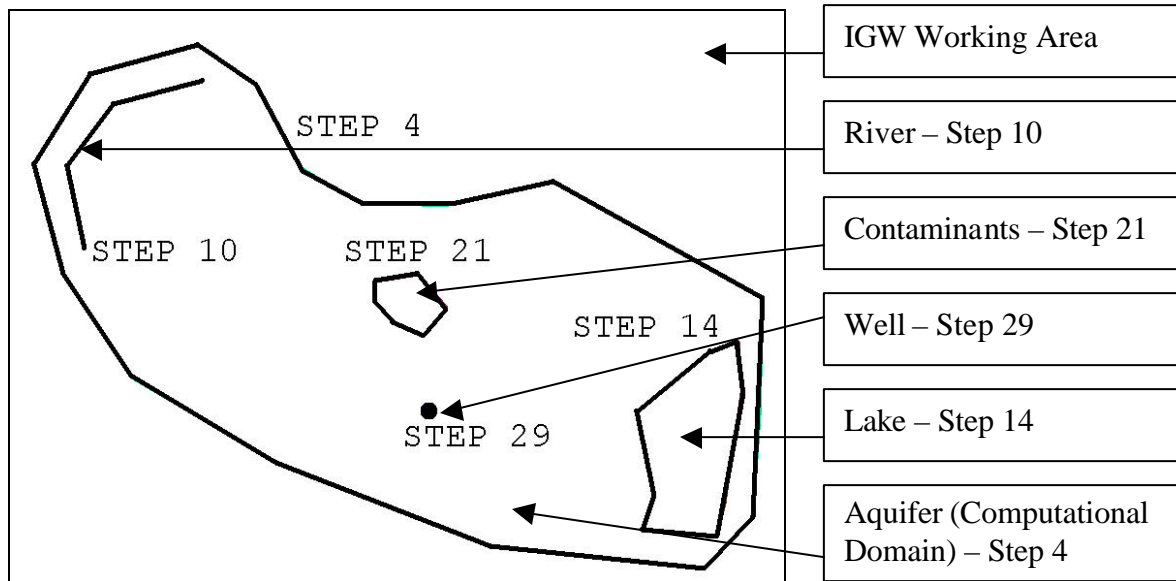


FIGURE 1
Sample of the Working Area

Step 1: Start the software (select 'Continue' when the splash screen appears).

Step 2: Examine the 'Tip of the Day' window (explore the options if desired). Close the window when finished.

The first modeling step is to create a computational domain.

Step 3: Click the 'Create a New Arbitrary Zone and Assign Property' button.



Step 4: Draw the desired shape of the polygon in the Working Area (white rectangular region) – see the polygon labeled 'Step 4' in **Figure 1**.

- Click the mouse to set a polygon vertex.
- Move the mouse to the next vertex location and click it again.
- Repeat a) and b) until each polygon vertex is defined.
- Double-click the mouse.

After defining a feature in the model, the user should assign its attributes.

Step 5: Press the 'Ctrl' key (or select 'Show model explore window' from the 'Display' menu).

This opens the 'Attribute Input and Model Explore' window (AIME) that is used to define the characteristics for each feature defined in the model.

'Zone 101' (default name) will be selected in the left-hand pane (LHP) of the AIME. Notice that this corresponds to the feature being outlined in red in the Working Area.

The right-hand pane (RHP) of the window shows the attributes for the feature currently selected in the LHP.

Step 6: Access the elevation data for the zone by clicking on the 'Aquifer Elevations' tab.

DR. SHUGUANG LI AND ASSOCIATES

INTERACTIVE GROUNDWATER MODELING 3 (IGW 3) QUICK TUTORIAL

This displays the 'Aquifer Elevations' layer of the 'Zone 101' RHP.

Step 7: Check the box next to 'Surface Elevation' and enter 0 in the appropriate field.

Notice the 'Apply' button in the upper right hand corner of the AIME. This button should be clicked when the user changes a value in a field (not including the first time).

Note that although only the surface elevation was defined, other attributes associated with the zone are defined and set in the 'Default Attribute' window (refer to the *IGW 3 User's Manual* for more information).

Step 8: Press the 'Ctrl' key to move the AIME (or right-click on the title bar of the window and select 'Move Form Quickly').

After the computational domain is defined, the next logical step is to define the sources and sinks in the model.

Step 9: Click the 'Create New Polyline and Assign Property' button.



Step 10: Draw a polyline in the upper-left hand corner of the Working Area, within the computational domain, to represent a river – see the line labeled 'Step 10' in **Figure 1**.

- Click the mouse to set one end of the polyline.
- Move the cursor to the line segment endpoint and click the mouse.
- Repeat b) for each line segment in the polyline.
- Place the cursor at the end of the final line segment and double-click the mouse.

Step 11: Access the AIME as in Step 5.

'Pline 102' (default name) will be selected in the LHP of the AIME and its attributes displayed in the RHP.

Step 12: Select 'Constant Head' and enter 5 in the field that appears.

Step 13: Move the AIME as in Step 8.

Step 14: Repeat Steps 3 and 4 to define a zone in the lower right-hand corner of the Working Area (within the computational domain) that will represent a lake – see the polygon labeled 'Step 14' in **Figure 1**.

Step 15: Access the AIME as in Step 5. 'Zone 103' will be selected in the LHP and its RHP will be displayed.

Step 16: Access the source / sink settings for the zone by clicking on the 'Sources and Sinks' tab.

Step 17: Click the 'River' checkbox and enter 2 in the 'Constant' field in the 'Stage' area. Note that the river selection can be used for any general head dependent source / sink.

With a computational domain and source / sinks defined, the model may now be solved.

Step 18: Click the 'Convert the Model into a Numerical Model' button. This discretizes the model and prepares it for the solving step.



Step 19: Click the 'Forward' button. The software solves the model and updates the working area to show the velocity vectors and the head contours.



Particles may be added to the model at any time.

Step 20: Click the 'Add Particles Inside a Polygon' button.



DR. SHUGUANG LI AND ASSOCIATES

INTERACTIVE GROUNDWATER MODELING 3 (IGW 3) QUICK TUTORIAL

Step 21: Define a polygon in the computational domain that will contain the particles – see the polygon labeled ‘Step 21’ in **Figure 1**. Refer to Step 4 for instructions on defining a polygon.

When the mouse is double-clicked the ‘Particles’ window will appear prompting for the number of columns of particles to be released within in the zone.

Step 22: Click the ‘OK’ button to accept the default value of 15.

The defined zone fills with particles.

Step 23: Click the ‘Forward’ button. The software will track the particles along the velocity vectors. Note that there is no need to discretize the model after adding particles.



Step 24: Click the ‘Pause’ button. The software finishes the current calculation and screen update and the stops the simulation.



Step 25: Click the ‘Backward Particle Tracking’ button. The will software will track the particles backward based on the velocity vectors.



Step 26: Stop the simulation as per Step 24.

Step 27: Click the ‘Delete All Particles’ button to remove the particles from the Working Area.



A powerful feature of the IGW 3 software is the ability to easily add or adjust features and quickly reformulate the model to obtain an updated solution.

Step 28: Click the ‘Add a New Well’ button.



Step 29: Click the mouse at a point in the computational domain – see the point labeled ‘Step 29’ in **Figure 1**. Do not make any adjustments in the AIME (the default settings will be used).

Step 30: Repeat Steps 18 and 19 to obtain a new model solution that incorporates the pumping well effects.

In addition to particles, IGW 3 can also model concentration plumes as a method for contaminant transport prediction.

Step 31: Define a zone as per Steps 3 and 4 in the same location as the particle zone – see the polygon labeled ‘Step 21’ in **Figure 1**.

Step 32: Access the AIME as in Step 5. ‘Zone 106’ will be selected in the LHP and its RHP will be displayed.

Step 33: Access the source / sink settings as per Step 16.

Step 34: Click the ‘Inst Conc.’ checkbox in the ‘Source Concentration’ area and enter 100 in the appropriate field.

Step 35: Discretize the model as per Step 18.

Step 36: Solve the model as per Step 19. The software will simulate the advection and dispersion of the contaminant plume.

Step 37: Stop the simulation as per Step 24.