# APPENDIX B

# STEP-BY-STEP APPLICATIONS OF THE FEMWATER-LHS

## Steady Two-Dimensional Drainage Problem

1. Double-click on the Argus ONE icon to open Argus ONE.
2. From the PIEs menu found along the top of the window, select, **New FEMWATER Project....** This brings up the *FEMWATER-LHS Type of Simulation Problem* window.
3. Here, the type of problem to be simulated is chosen. To select the type of **CROSS-SECTIONAL** click on check box and then click **Continue.** The *FEMWATER-LHS Model* dialog appears.
4. This window allows the user to specify values for the FEMWATER simulation that are not spatially variable. This dialog can be get again at any time by selecting **PIEs|Edit Project Info**. Rather than making changes here now, accept the default values by clicking **OK**. This brings up a new *Argus* ONE *window,* called "untitled1".
5. This is the window in which the model will be designed, run, and evaluated. It contains many layers in a stack; each layer will hold either model or mesh information. Additionally, another window (Fig. B.1), the *Layer List window* (also called "Layer Floater") appears, in which the user can see which layers are available. This window may be resized to display the full layer names.

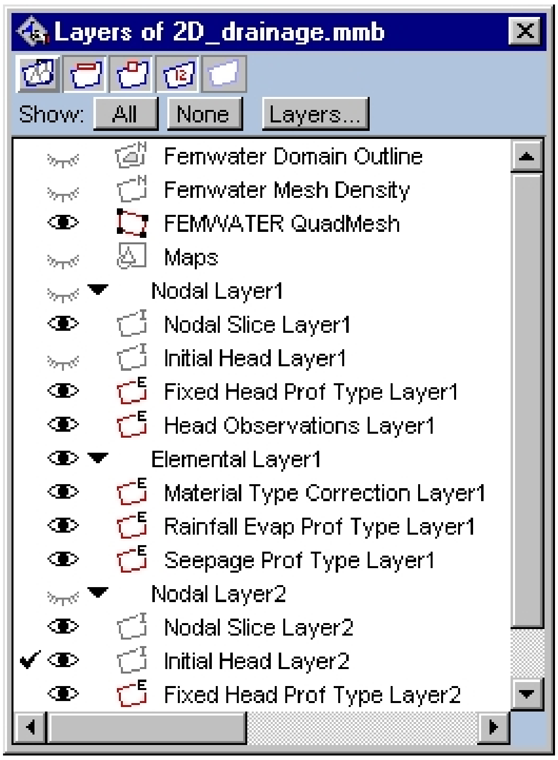


Fig. B.1 *Layer List Window*

The *Layer List window* shows which information layers are available for the particular problemtype, i.e. **Nodal Elevation Layer[i] or Nodal Slice Layer[i], Initial Head Layer[i], Material Type Correction Layer[i]**, etc. The window allows the user to control which of the layers will be visible (those with the open eye) and which layer is on top of the stack and thus available for input from the screen. Clicking on an 'eye' toggles the layer visibility, and clicking to the left of an 'eye' makes the layer 'active' (i.e. brings the layer to the top of the stack) and puts a 'check mark' next to the active layer.

6. To create a rectangular uniform elements. It is possible to read a ***Grid*** (rectangular uniform elements) into ***FEMWATER Quadmesh*** directly. However, mesh BandWidth can be minimized once the mesh is imported. A text file (e.g. *2D_drainage_mesh.exp*) containing 100 elements and 121 nodes was used to represent the flow region. Activate the ***FEMWATER Quadmesh*** by clicking to the left of its 'eye' in the *Layer List window.* Import *2D_drainage_mesh.exp* into the project by selecting **File|Import FEMWATER QuadMesh...|Text File**. In CD-ROM, the *2D_drainage_mesh.exp* file is located in a directory with the pathname examples\application_1\app1_import_files\.
7. To specify a constant slice position, or default slice position for the nodal of layer, the **Layers dialog** must be used. Moving the cursor to the *Layers...* button in the floating layers window and clicking, opens the Layers dialog. The list at the top of the dialog is the list of layers. Highlighting the layer under consideration, in this case **Nodal Slice Layer2**, in that list by clicking it with the cursor shows the parameters associated with that information layer in the table at the bottom of the dialog box. Moving the cursor to the **Value** column and clicking ***fx*** the *expression* box to appear. Just type 10 in the *expression* box and clicking **OK** exits the *expression* dialog.
8. To modify the non-spatial data (i.e. boundary conditions profile types and material correction types) in this project. Select **PIEs|Edit Project Info**, the *FEMWATER-LHS Model* dialog will appear.
9. Click on *Material and Soil Properties* tab to activate material correction by clicking *Material type correction* check box and set the following parameters:

   On *Correction Cond/Perm* tab, set

| N | xx | yy | Zz | xy | xz | yz |
|---|---|---|---|---|---|---|
| 1 | 0.06 | 0.0 | 0.06 | 0.0 | 0.0 | 0.0 |

   On *Correction Soil Prop* tab, set

| N | Res MC | Sat MC | P Head | VG Alpha | VG Beta |
|---|---|---|---|---|---|
| 1 | 0.034 | 0.046 | 0.0 | 1.6 | 1.37 |

10. Click on the *Boundary Conditions|Dirichlet* tab to activate fixed head profile type by clicking *Fixed Head* check box. Enter the values of the profile. Specify 2 m for the head in unlimited time, enter the same values of **Head 1** and **Head 2** with 2, **Time 1** with 0 and **Time 2** with the 1.0e38.
11. Click on the *Boundary Conditions|Variable Composite* tab to activate rain fall and seepage profile type by clicking *Rainfall/Evap-Seepage* check box. Enter the values of the first profile. Specify 0.006 m/day for the rain fall in unlimited time, enter the same values of **Rf/Evap 1** and **Rf/Evap 2** with 0.006, **Time 1** with 0 and **Time 2** with the 1.0e38. Add the number of profiles by clicking on the **Add Rows** button. Enter the values of the second profile. There are the same values of **Rf/Evap 1** and **Rf/Evap 2** with 0.0 in

unlimited time (**Time 1** with 0 and **Time 2** with the 1.0e38). Thus, there are two types (number 1 and number 2) rain fall and see-page boundary conditions. Finally clicking **OK** to finish the changes.

12. To enter the Dirichlet boundary conditions into the model, activate the ***Fixed Head Prof Type Layer1***, by clicking on its 'eye' in the *Layer List window*.
13. Draw a line by first activating the contour-drawing tool. To do this, click on the small quadrilateral just below the arrow along the left side of the *Argus* ONE *window* and select the Open Contour Tool from the pop-up menu. It is the middle item. Now draw a vertical line through the nodes in bottom left of the model and assign it a **Fixed Head Prof Type** to 1.
14. To set the precise node positions using *EditContours*. Select **File|Import Femwater Domain Outline|Edit Contours**. Then select the ***Fixed Head Prof Type Layer1*** from the list of layers. The objects on the ***Fixed Head Prof Type Layer1*** will be imported into the *EditContours* PIE. Click on any node there to select it and edit it's position (Fig. B.2).

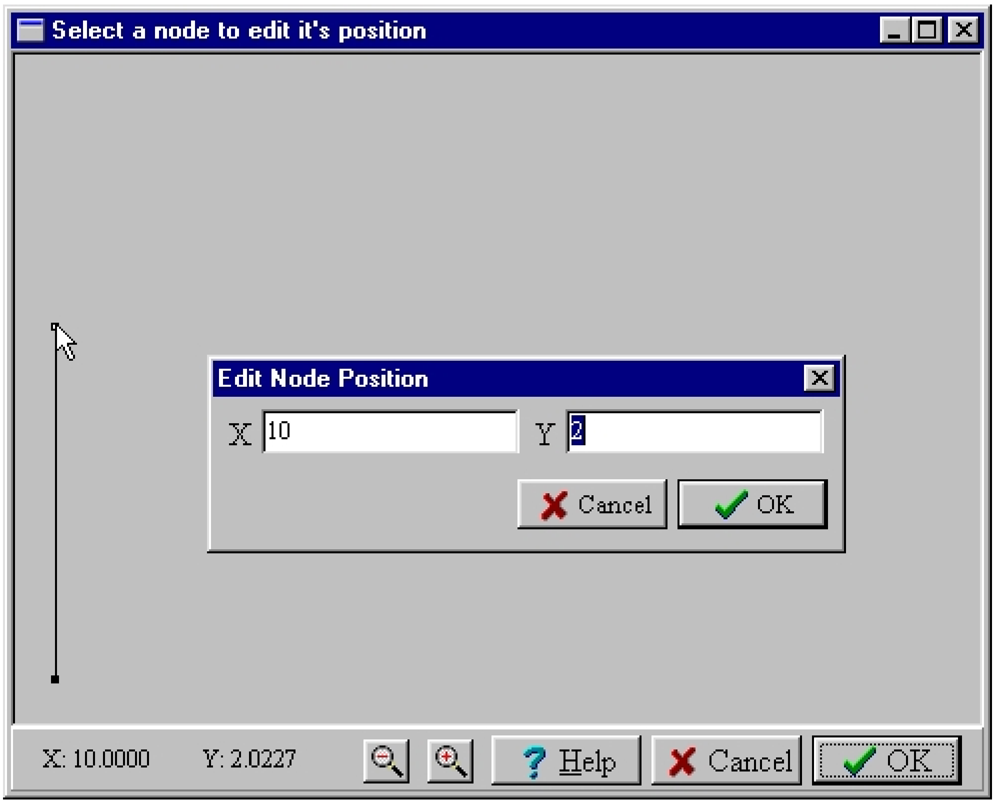


Fig. B.2 *EditContours* dialog

15. Copy this contour by pressing **Ctrl+C** or select **Edit|Copy.**
16. Activate the ***Fixed Head Prof Type Layer2***, by clicking on its 'eye' in the *Layer List window*. Select **Edit|Paste** to create this contour. Paste in the copied object by pressing **Ctrl+V** (or select **Edit|Paste**).
17. To enter the Rainfall/Evaporation boundary conditions into the model, activate the ***Rainfall Evap Prof Type Layer1***, by clicking on its 'eye' in the *Layer List window*.

18. Click on the small quadrilateral just below the arrow along the left side of the *Argus* ONE *window* and select the Open Contour Tool from the pop-up menu. It is the middle item. Draw a horizontal line through the top row elements and set:

    **Rainfall Evap Prof Type** = 1
    **Ponding Depth** = 0
    **Min Pressure Head** = -9000

    and click **OK**.

19. To enter the Seepage boundary conditions into the model, activate the ***Seepage Prof Type Layer1***, by clicking on its 'eye' in the *Layer List window*.
20. Draw a vertical line through the 8 elements on the left edge of the model and set:

    **Seepage Prof Type** = 2
    **Ponding Depth** = 0
    **Min Pressure Head** = -9000

    and click **OK**.

21. To enter the material correction type into the model, activate the ***Material Type Correction Layer1***, by clicking on its 'eye' in the *Layer List window*.
22. Click on the small quadrilateral just below the arrow along the left side of the *Argus* ONE *window* and select the Close Contour Tool from the pop-up menu. It is the first item. Draw a polygon through the 3 rows of element on the top of the model and assign it a **Material Type** to 1.
23. Save the project so far by clicking **File**, and then **Save As...**. Select the desired directory and type in the desired name (e.g. *2D_drainage*) and then click on **Save**. A project file called *2D_drainage.mmb* is created in the directory you chose, and the window name becomes the same, as shown in Fig. B.3.
24. The model information entered now needs to be exported from Argus ONE creating input files that FEMWATER-LHS requires, and the simulation can then be run. (Note that the ***FEMWATER QuadMesh*** layer must be active in order to export.) In the **PIEs** menu, select **Run FEMWATER**. The **Run FEMWATER** dialog box appears.

    The full paths to the executables should be displayed in edit-boxes on the FEMWATER Path the **Run FEMWATER** dialog box. If the executable for the chosen model is not at the location specified in the edit-box, the background of the edit-box and the status bar will change to red and a warning message will be displayed in the status bar to indicate that the path is incorrect. Normally, the user should correct the path before attempting to create the input files. Although it is possible to export the input files using an incorrect path. *Argus* ONE will not be able to start the model if the path is incorrect. Type the correct path or click on the Browse button to set the correct path. When a model is saved, the paths for all of the models will be saved in a file named *Femwater_lewaste_lhs.ini* in the directory containing the FEMWATER PIE. *Femwater_lewaste_lhs.ini* will be read whenever a new FEMWATER project is created or an old one is read so that the model paths do not need to be reset frequently. In this windows also allows the user to choose only creation of FEMWATER-LHS input files, or both creation of files and running of FEMWATER-LHS (which is already selected). Click **OK** to proceed. An *Enter export file name window* appears.

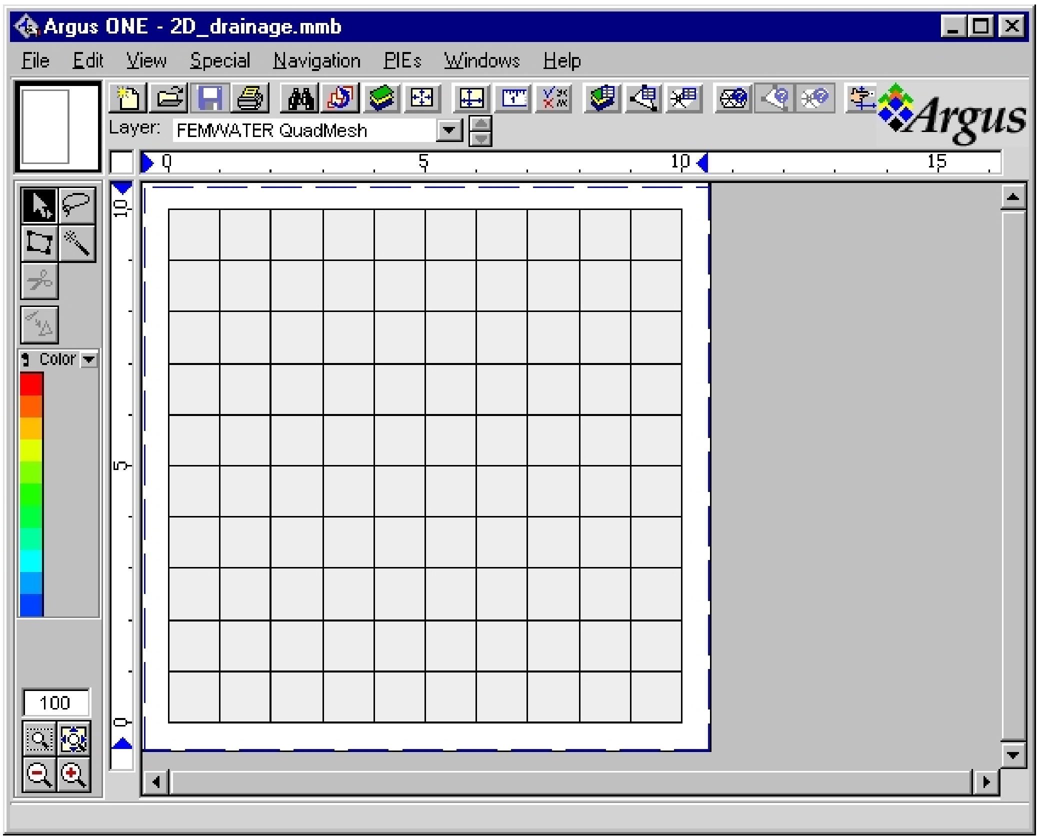


Fig. B.3 FEMWATER Mesh in Steady Two-Dimensional Drainage Problem

25. Select the directory into which the FEMWATER-LHS input files will be placed by Argus ONE. Then select the name of the files by typing in the space next to **File Name** (e.g. *2D_drainage*). The files created will all begin with the name entered here. Note, ignore the **Save as type** box.
26. Click on **Save** and the export takes place while the barber pole is visible, and then the FEMWATER-LHS simulation is run while the DOS window is visible.
27. To visualize the results, select **PIEs|FEMWATER Post Processing**. The Select Data Set window appears (Fig. B.4). Chose the type of FEMWATER output file to be read by clicking on *FEMWATER (Head)* check box, and click **Select Data Set**. Find the directory selected above for the FEMWATER files and double-click the appropriate "*.hef" file (e.g. *2D_drainage.hef*). This brings up the *FEMWATER Post-Processing* window (Fig. B.5). This window contains a list of all results available from FEMWATER simulation for visualization. Because the simulation was steady-state conditions, only one time step appears, select **Contour Map** from the list of chart types. Then click **OK** and the plots are created.
28. Because the ***FEMWATER QuadMesh*** layer was active, the plots appear below the mesh. Bring the plots to the top of the stack by activating the ***FEMWATER Post Processing Charts*** layer click left of the 'eye' in the *Layer List window*).

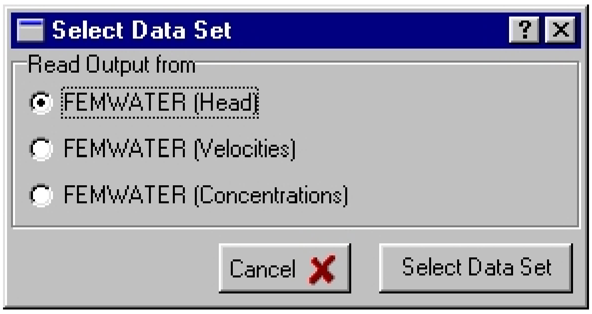


Fig. B.4 Select Data Set dialog.

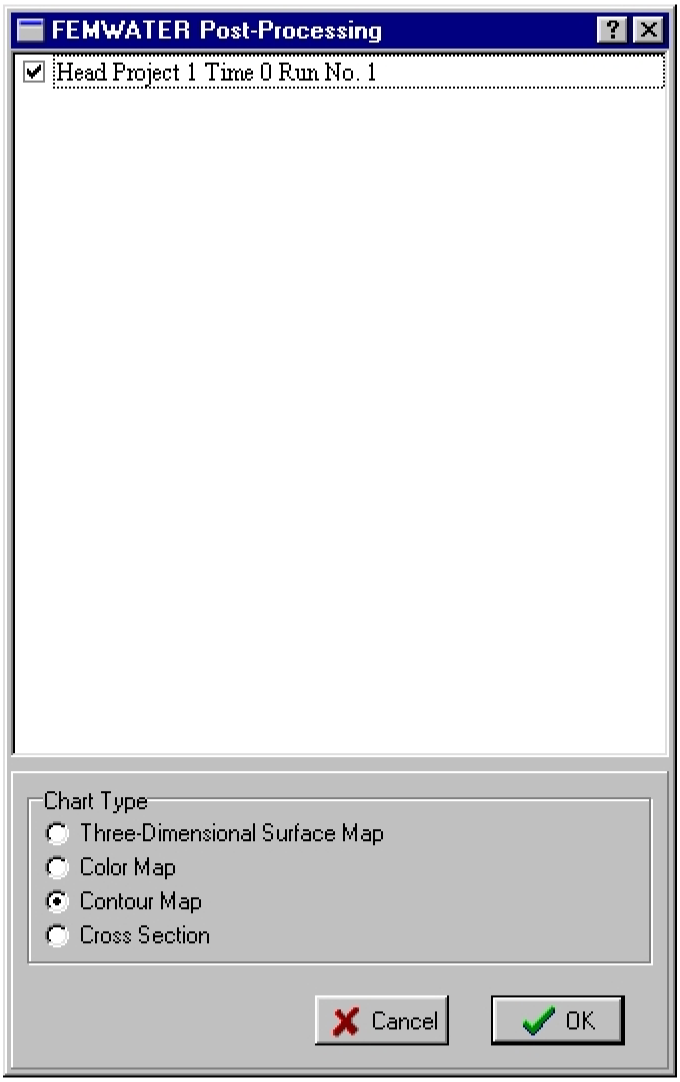


Fig. B.5 FEMWATER Post Processing dialog.

29. The plot appears, but is too cluttered because the mesh is also visible. Make the mesh invisible by clicking on the 'eye' next to ***FEMWATER QuadMesh*** in the *Layer List window.* A plot of head contours, similar to that in Fig. B.6 is visible.
30. Save the current state of the project by selecting **File**, and then **Save**.

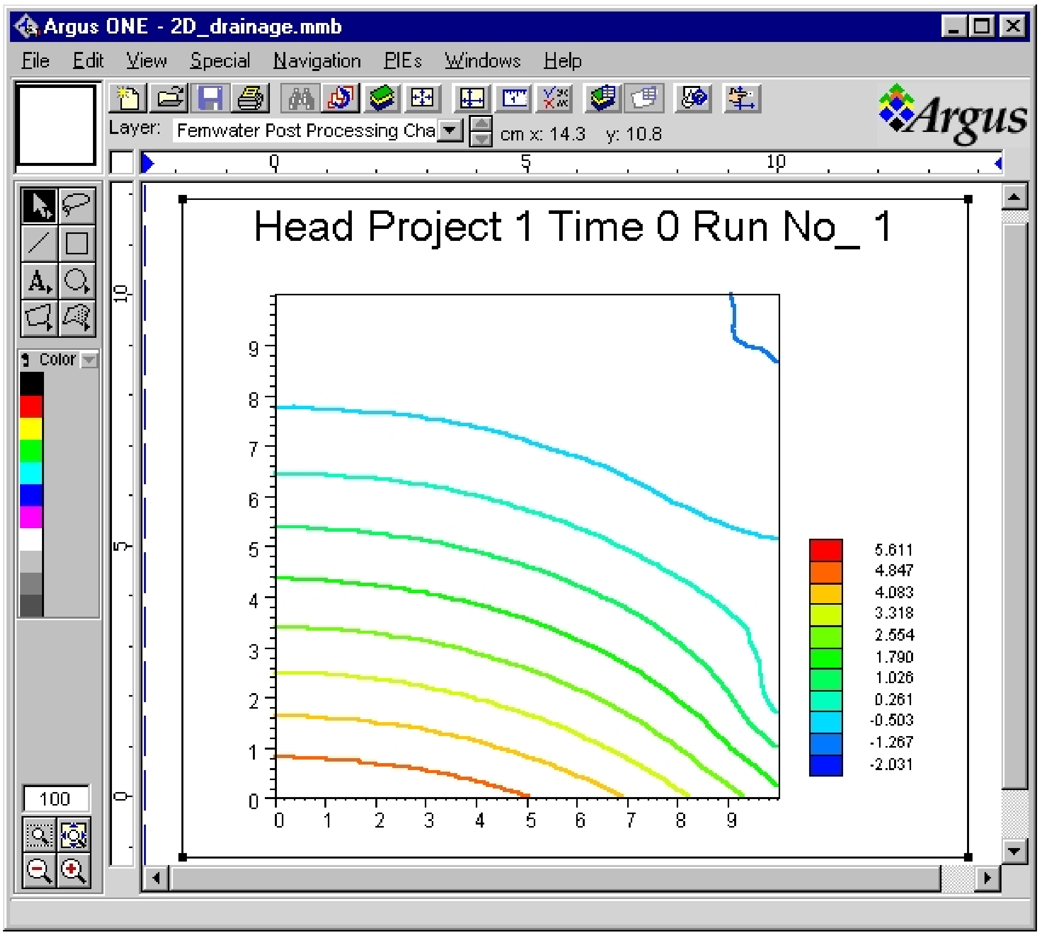


Fig. B.6 Head Plot in Steady Two-Dimensional Drainage Problem

31. The Argus ONE application may now be closed by selecting **File**, and then **Quit**. The same state that the project was left in will be reproduced when the project is reopened in Argus ONE at any later time.

### Steady Two-Dimensional Drainage Problem (LHS mode)

1. Double-click on the Argus ONE icon to open Argus ONE.
2. Select **File**, then **Open...,** to bring back the project that was saved in the above case. In the *Choose file to open: window* that appears, move to the appropriate directory and double click on the ".mmb" project file that was saved in the above case (e.g. *2D_drainage.mmb*). This returns the user environment to the same state as when the project was previously saved (Fig. B.6).
3. To set this simulation, the FEMWATER-LHS Non-Spatial Information must be modified. Brings up this dialog by using **PIEs**, then **Edit Project Info ....**
4. On *Model Title and Type* tab, change the Project Number by setting **NPROB** = 2 and click on *Latin Hypercube Sampling Simulation* check box, to change the type of simulation.
5. Click on the tab along the top of dialog that read, *Latin Hypercube Sampling*. Set the **Number of Runs** to 50. In order that the simulation will be run 50 times with the 50 sets of variable.

6. To enter the data directly in the table, click on the *Variables* tab (Fig. B.7). For *Variable Name, Correlated, Material Type* and *Distribution Type* select a cell and then click in it or press the **Enter** key on the keyboard to display a list of choices. For the other columns, just select the cell and type the data. The **Add** button can be used to add a new variable to the end of the list of variables. To delete a variable from the list, the cursor is moved to the table listing the variables. Clicking the row in the table highlights the information and then clicking the **Delete** button removes the variable from the list. Clicking the **Insert** button adds a variable below the highlighted variable. In this case assume that all soil hydraulic parameters for the two material types (e.g. default and correction(1)) have a

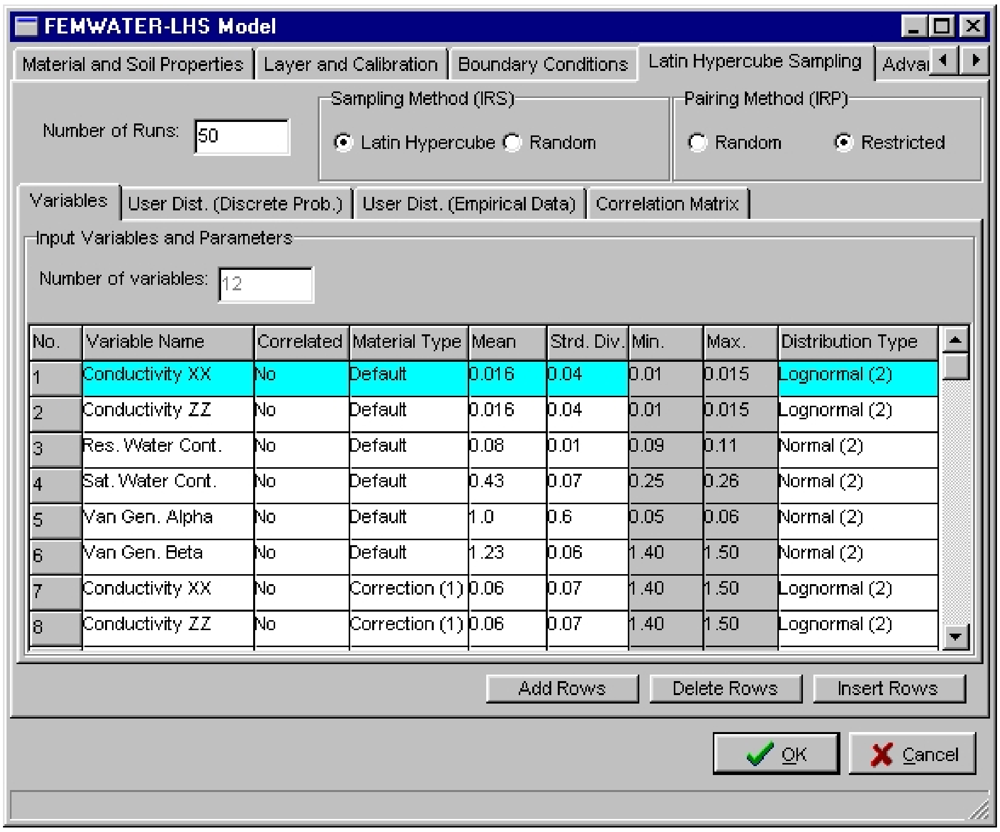


Fig. B.7 *Latin Hypercube Sampling* tab dialog

probabilistic distributions. Set the following parameters:

| N | Variable Name | Correlated | Mat. Type | Mean | Std Dev | Distribution Type |
|---|---|---|---|---|---|---|
| 1 | Conductivity XX | No | Default | 0.0168 | 0.0456 | Lognormal (2) |
| 2 | Conductivity YY | No | Default | 0.0168 | 0.0456 | Lognormal (2) |
| 3 | Res. Water Cont. | No | Default | 0.089 | 0.01 | Normal (2) |
| 4 | Sat. Water Cont. | No | Default | 0.43 | 0.07 | Normal (2) |
| 5 | V. G. Alpha | No | Default | 1 | 0.6 | Normal (2) |
| 6 | V. G. Beta | No | Default | 1.23 | 0.06 | Normal (2) |
| 7 | Conductivity XX | No | Correction (1) | 0.06 | 0.0792 | Lognormal (2) |
| 8 | Conductivity YY | No | Correction (1) | 0.06 | 0.0792 | Lognormal (2) |
| 9 | Res. Water Cont. | No | Correction (1) | 0.034 | 0.01 | Normal (2) |
| 10 | Sat. Water Cont. | No | Correction (1) | 0.46 | 0.11 | Normal (2) |
| 11 | V. G. Alpha | No | Correction (1) | 1.6 | 0.7 | Normal (2) |
| 12 | V. G. Beta | No | Correction (1) | 1.37 | 0.05 | Normal (2) |

7. Click **OK** to exit the dialog. Save the project (e.g. to 2D_drainage_lhs.mmb).
8. To export and run, activate the ***FEMWATER QuadMesh*** layer again bringing the mesh to the top of the stack. Select **PIEs**, and then **Run FEMWATER**.
9. Click **OK** in the *Run FEMWATER window*, and Select the directory into which the FEMWATER-LHS input files will be placed and select the name of the new files that will run the LHS mode simulation (e.g. *2D_drainage_lhs*). Note, ignore the **Save as type** box. Click on **Save** to export and run.
10. The results can be analyzed in statistically. However, the results can be visulaized. From the **PIEs** menu select **FEMWATER Post Processing**. Click on **FEMWATER (Head)** check box in the select data file dialog and click **Select Data Set**. Find the directory selected above for the FEMWATER files and double-click the appropriate "*.hef" file (e.g. *2D_drainage_lhs.hef*). The *FEMWATER Post-Processing* window will be appear. Select the data file and select *Contour Map* from the list of chart types. Then click **OK** and the plots are created.

### Transient Two-Dimensional Drainage Problem

1. Double-click on the Argus ONE icon to open Argus ONE.
2. Select **File**, then **Open...,** to bring back the project that was saved in the above case. In the *Choose file to open: window* that appears, move to the appropriate directory and double click on the ".mmb" project file that was saved in the above case (e.g. *2D_drainage.mmb*). This returns the user environment to the same state as when the project was previously saved.
3. In this case the region consist only one material type. There is no material correction exist. To modify this information, brings up the FEMWATER-LHS Non-Spatial Information dialog by using **PIEs**, then **Edit Project Info ...**.
4. On *Model Title and Type* tab, set **NPROB =** 3.
5. To change the type of solution, click on *Run Control* tab and click *Transient State Solution* check box.
6. The transient simulation will be performed for 50 time steps. The initial time step size is 0.25 day and each subsequent time step size is increased with a multiplier of 2.0 with the maximum time step size of less than or equal to 32 days. The pressure head tolerance for nonlinear iteration is $2 \times 10^{-3}$ m. The relaxation factor for the nonlinear iteration is set equal to 0.5. To input this information, click on *Time Control* tab and set:

   **NTI** = 50
   **TMAX** = 2000
   **DELT** = 0.25
   **CHNG** = 2.0
   **DELMAX** = 32

   and click **OK**.

7. To delete or inactivate the material correction, click on *Material and Soil Properties* tab and then clicking *Material type correction* check box.
8. In this simulation the initial conditions were used the steady state solution resulting from zero flux on the top. However, this solution is already done that so it can be just import those contours. Activate the ***Initial Head Layer1***, by clicking on its 'eye' in the *Layer List*

*window*. Import *2D_drainage_init.exp* into the project by selecting **File|Import Initial Head Layer1|Text File**, and then select *2D_drainage_init.exp*.

9. Copy this contour by pressing **Ctrl+C** or select **Edit|Copy.**
10. Activate the ***Initial Head Layer2***, by clicking on its 'eye' in the *Layer List window*. Select **Edit|Paste** to create this contour. Paste in the copied object by pressing **Ctrl+V** (or select **Edit|Paste**).
11. Save the project (e.g. to *T_2D_drainage.mmb*).
12. Export and run FEMWATER-LHS.
13. Then, from the **PIEs** menu, select **FEMWATER Post Processing** and then reselect the "hef" file (e.g. *T_2D_drainage.hef*). There are 50 time steps, but choose only the last time step, Time 2852.4 Select **Contour Map** and click the **OK**. A plot of head contours, similar to that in Fig. B.8 is visible.

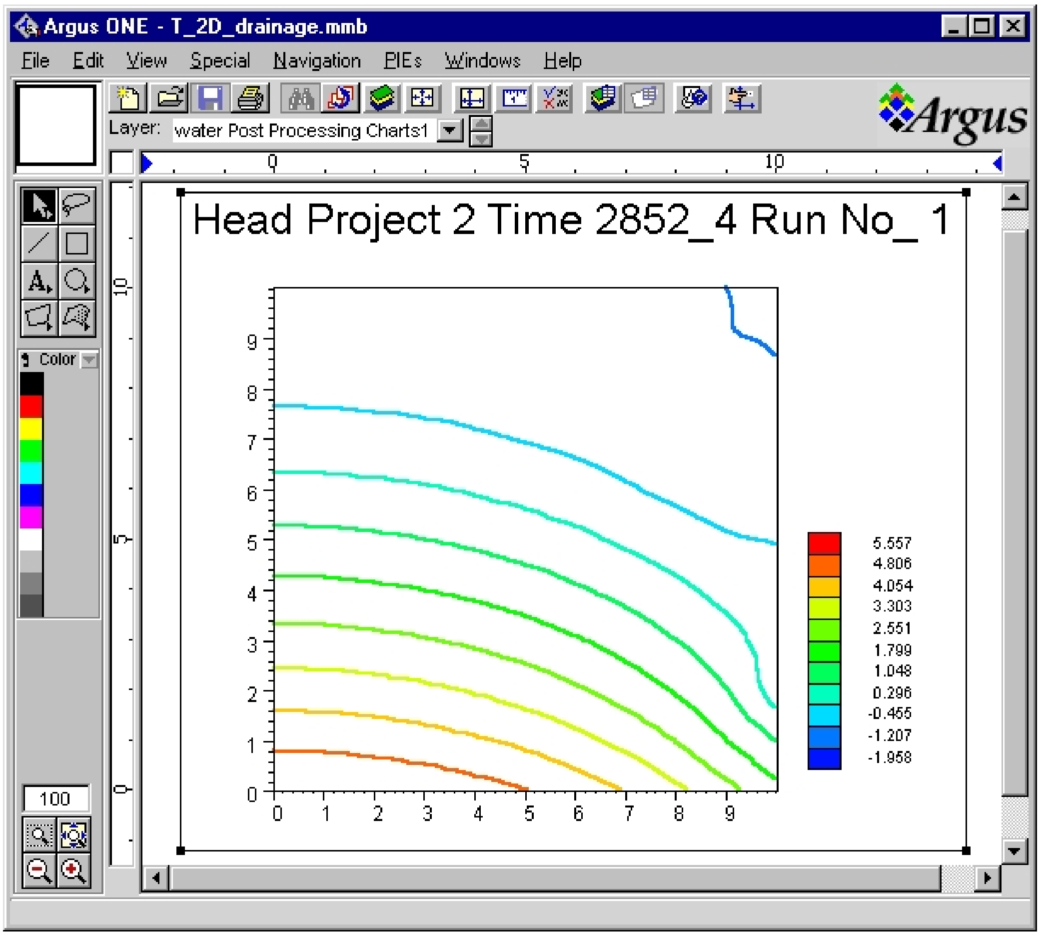


Fig. B.8 Head Plot in Transient Two-Dimensional Drainage Problem

### Transient Two-Dimensional Drainage Problem (LHS mode)

1. Start Argus ONE. Open the ".mmb" project file that was saved in the above case (e.g. *T_2D_drainage.mmb*).

2. Select **PIEs**, then **Edit Project Info ...** to modify the the FEMWATER-LHS Non-Spatial Information.

   On *Model Title and Type* tab, set **NPROB** = 4 and click *Latin Hypercube Sampling Simulation* check box

   On *Latin Hypercube Sampling* tab, set the **Number of Runs** to 32

   On *Latin Hypercube Sampling|Variables* tab, set the following parameters:

| N | **Variable Name** | **Correlated** | **Mat. Type** | **Mean** | **Std Dev** | **Distribution Type** |
|---|---|---|---|---|---|---|
| 1 | Conductivity XX | No | Default | 0.0168 | 0.0456 | Lognormal (2) |
| 2 | Conductivity YY | No | Default | 0.0168 | 0.0456 | Lognormal (2) |
| 3 | Res. Water Cont. | No | Default | 0.089 | 0.01 | Normal (2) |
| 4 | Sat. Water Cont. | No | Default | 0.43 | 0.07 | Normal (2) |
| 5 | V. G. Alpha | No | Default | 1 | 0.6 | Normal (2) |
| 6 | V. G. Beta | No | Default | 1.23 | 0.06 | Normal (2) |

   Then click **OK**.

3. Save the project (e.g. to T_2D_drainage_LHS.mmb).
4. Export and run FEMWATER-LHS.
5. Uncertainty and sensitivity analysis of the input and the output variables.

### Steady Three-Dimensional Pumping Problem

1. Start Argus ONE. In the *FEMWATER-LHS Type of Simulation Problem* window, select a **Areal** orientation. Then click **Continue**.
2. In the FEMWATER-LHS dialog, a number of changes to the initial default values are required.

   On *Model Title and Type* tab, set **NPROB** = 5

   On *Material and Soil Properties* tab, click *Material type correction* check box and click *Add Rows* button to add the number of correction material. Set the following parameters:

   On *Cond/Perm* tab, set

| **xx** | **yy** | **zz** | **xy** | **xz** | **yz** |
|---|---|---|---|---|---|
| 0.3144 | 0.3144 | 0.3144 | 0.0 | 0.0 | 0.0 |

   On *Soil Prop* tab, set

| **Res MC** | **Sat MC** | **P Head** | **VG Alpha** | **VG Beta** |
|---|---|---|---|---|
| 0.1 | 0.39 | 0.0 | 5.8 | 1.48 |

   On *Correction Cond/Perm* tab, set

| N | **xx** | **yy** | **zz** | **xy** | **xz** | **yz** |
|---|---|---|---|---|---|---|
| 1 | 1.0608 | 1.0608 | 1.0608 | 0.0 | 0.0 | 0.0 |
| 2 | 7.128 | 7.128 | 7.128 | 0.0 | 0.0 | 0.0 |

On *Correction Soil Prop* tab, set

| N | Res MC | Sat MC | P Head | VG Alpha | VG Beta |
|---|---|---|---|---|---|
| 1 | 0.065 | 0.41 | 0.0 | 7.5 | 1.89 |
| 2 | 0.045 | 0.43 | 0.0 | 14.5 | 2.68 |

On *Layer and* Calibration tab, click **Add** button 9 times to set the number of elemental layers to 10 or the number of nodal layers to 11.

On the *Boundary Conditions|Dirichlet* tab, click *Fixed Head* check box and then click **Add Rows** button to add the number of profile. Set the profiles as follows:

| N | Time 1 | Head 1 | Time 2 | Head 2 |
|---|---|---|---|---|
| 1 | 0.0 | 60.0 | 1.0e38 | 60.0 |
| 2 | 0.0 | 30.0 | 1.0e38 | 30.0 |

3. In the *Argus* ONE *window*, select **Special|Scale and Units...** and set **Uniform Scale:** = 50. That it reads "Every 1 cm on the screen represents 50 units in the real world in both the x and y direction".
4. Activate the ***Femwater Domain Outline*** layer and draw a model boundary with the contour-drawing tool. Try to create a square outline with 1000 m wide and 400 m high. Then, double-click on the location desired for the last vertex. The *Contour Information dialog* appears. Here the desired typical size of finite elements to be created by the mesh generator is specified. Type 40 in the space below the label, **Value**. This sets the desired width of an element to 40 in the units shown in the rulers around the periphery of the workspace. Click **OK** to exit the window.
5. To set the precise node positions using *EditContours*. Select **File|Import Femwater Domain Outline|Edit Contours**. Then select the ***Femwater Domain Outline*** layer from the list of layers. The objects on the ***Femwater Domain Outline*** layer will be imported into the *EditContours* PIE. Click on any node there to select it and edit it's position.
6. To copy the boundary and to convert it to an open contour. Activate the ***Femwater Domain Outline*** layer. Then use the lasso tool to outline all the cells that define where the constant head boundary ought to be (i.e. the left two cells). The selected cells will change from black squares to hollow squares. Copy them to the clipboard (**Edit|Copy**). Activate the ***Fixed Head Prof Type Layer1*** and paste the copied object by select **Edit|Paste**. A single open contour where the constant head boundary should be. Double click on it to bring up the *Contour Information dialog*. Set **Fixed Head Prof Type** = 1. Click **OK**.
7. Repeat step 6 to copy the other open contour (i.e. the right two cells) in ***Fixed Head Prof Type Layer1***.
8. To copy all open contours in constant head boundary layers (i.e. ***Fixed Head Prof Type Layer2*** through ***Fixed Head Prof Type Layer11***). Select **Edit|Select All** and then select **Edit|Copy**. Activate the ***Fixed Head Prof Type Layer2*** and select **Edit|Paste**. Paste the copied object to the remaining layers (i.e. ***Fixed Head Prof Type Layer3*** through ***Fixed Head Prof Type Layer11***).

9. Copy object in ***Femwater Domain Outline*** layer into ***Material Type Correction Layer6***. Double click on it to bring up the *Contour Information dialog*. Set **Material Type** = 1. Click **OK**.
10. Copy this object into ***Material Type Correction Layer7*** and ***Material Type Correction Layer8***.
11. Again copy this object into ***Material Type Correction Layer9*** and ***Material Type Correction Layer10***. But set those **Material Type** = 2.
12. Activate the ***Fixed Head Prof Type Layer1***. Click on the **Closed Contour** button and hold the mouse button down until a pop-up menu appears. Select the bottom selection which is the point tool. Click in the center of the model. A *Contour Information dialog* will appear. Set **Fixed Head Prof Type** = 2. Click **OK**.
13. Set the precise node positions of this point (i.e. **X** = 540 and **Y** = 400) using *EditContours*.
14. Copy this point into ***Fixed Head Prof Type Layer2*** and ***Fixed Head Prof Type Layer3.***
15. To force this point on the node, copy it into ***Femwater Domain Outline*** layer. Double click on it to bring up the *Contour Information dialog*. Set **elemen_size** = 40. This sets the desired width of an element to 40 in the units shown in the rulers around the point. Click **OK**.
16. Activate the ***FEMWATER QuadMesh*** layer. Click on the 'magic wand' found along the left side of the *Argus* ONE *window* near the arrow, and then click the magic-wand cursor inside the model boundary just drawn. An irregular finite element mesh containing elements of about 40 size is generated and displayed. Fig. B.9 shows the type of mesh that may be expected.

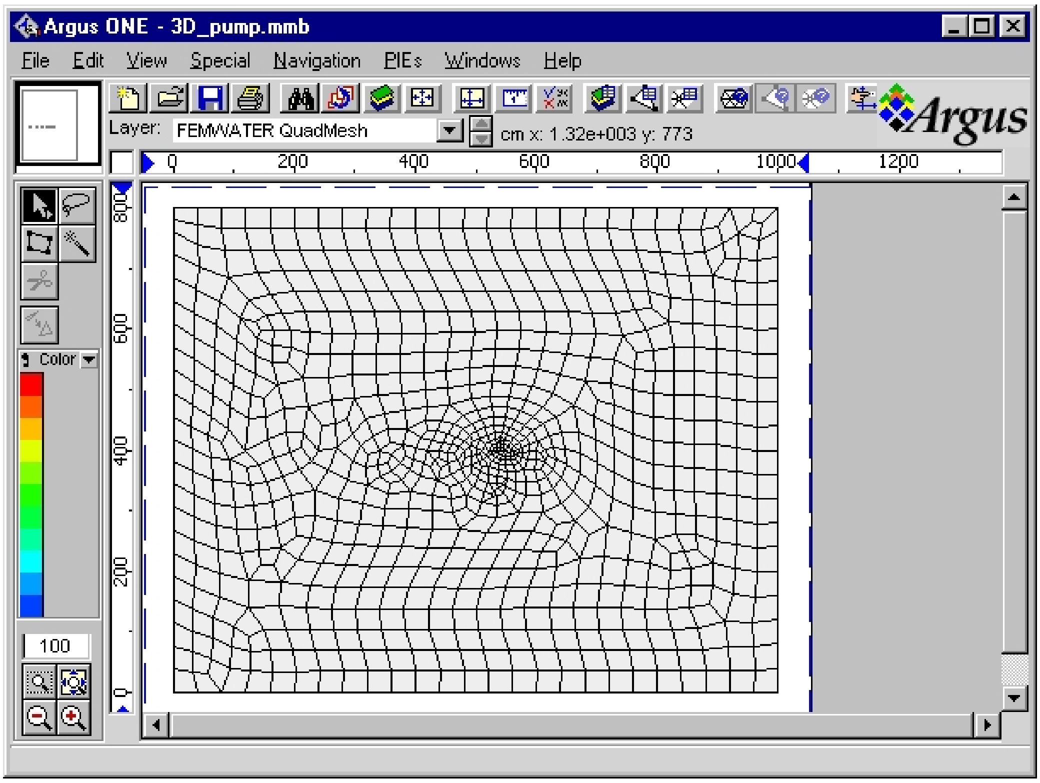


Fig. B.9 FEMWATER Mesh in Steady Three-Dimensional Pumping Problem

17. The band-width of a newly-generated mesh always needs to be reduced. Select the **Special** menu long the top of the *Argus* ONE *window*, then select **Renumber...**. This brings up the **Renumber window**. In this window click on **Optimize Bandwidth** and then **OK**. The mesh numbering is then optimized for the matrix solver currently used by FEMWATER-LHS.
18. Specify a constant elevation, or default elevation using the **Layers dialog**. In this case, enter 15, 30, 35, 40, 45, 50, 55, 60, 66, 72 in the **Expression dialog** for **Nodal Elevation Layer2** through **Nodal Elevation Layer11** respectively.
19. Specify a constant initial head, using the **Layers dialog**. In this case, enter 60, 45, 30, 25, 20, 15, 10, 5, 0, -6, -12 in the **Expression dialog** for ***Initial Head Layer1*** through ***Initial Head Layer11*** respectively.
20. Save the project (e.g. to *3D_pump.mmb*).
21. Export and run FEMWATER-LHS.
22. Using **FEMWATER Post Processing** in **PIEs** menu, plot pressure head for *Three-Dimensional Surface Map* type (Fig. B.10).

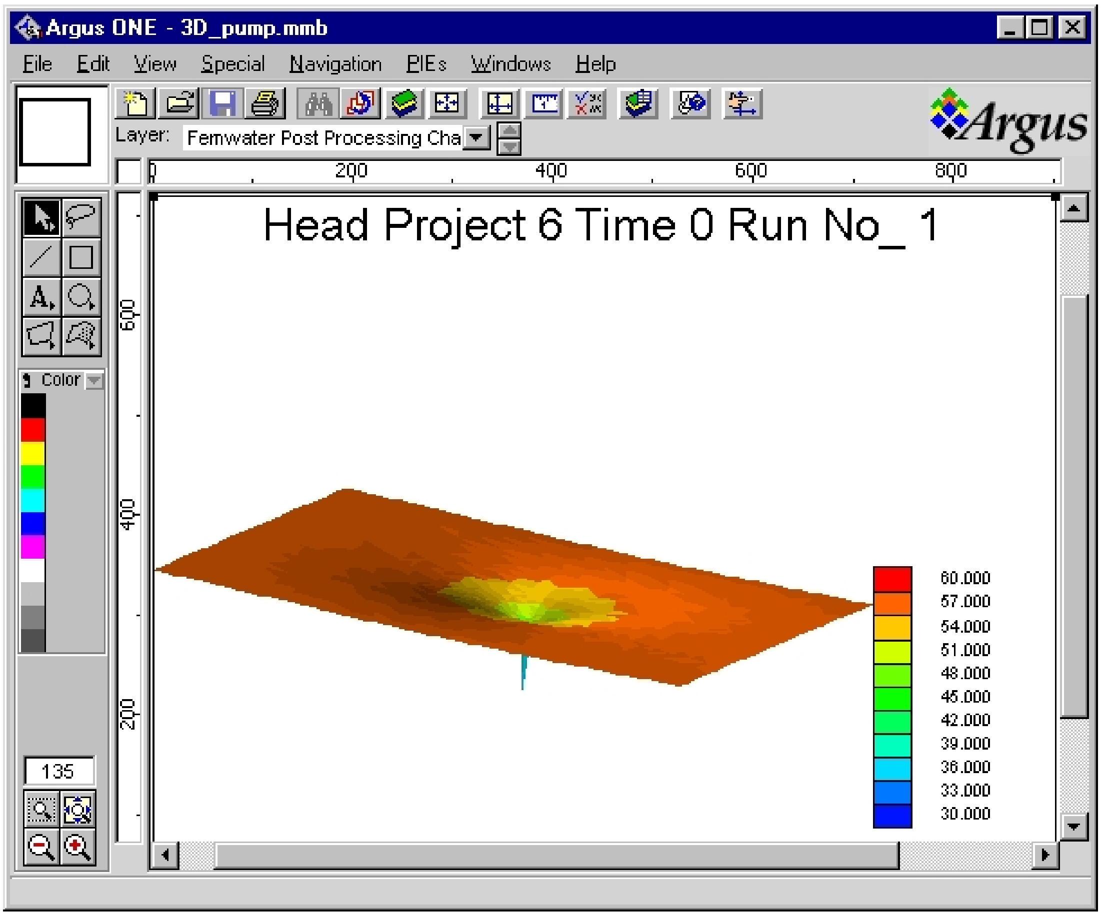


Fig. B.10 Head Surface Map in Steady Three-Dimensional Pumping Problem

### Steady Three-Dimensional Pumping Problem (LHS mode)

1. Start Argus ONE. Open the ".mmb" project file that was saved in the above case (e.g. *3D_pump.mmb*).
2. Select **PIEs**, then **Edit Project Info ...** to modify the the FEMWATER-LHS Non-Spatial Information.

   On *Model Title and Type* tab, set **NPROB** = 6 and click *Latin Hypercube Sampling Simulation* check box

   On *Latin Hypercube Sampling* tab, set the *Number of Runs* to 50

   On *Latin Hypercube Sampling|Variables* tab, set the following parameters:

| N | **Variable Name** | **Correlated** | **Mat. Type** | **Mean** | **Std Dev** | **Distribution Type** |
|---|---|---|---|---|---|---|
| 1 | Conductivity XX | No | Default | 0.3144 | 0.6576 | Lognormal (2) |
| 2 | Conductivity YY | No | Default | 0.3144 | 0.6576 | Lognormal (2) |
| 3 | Conductivity ZZ | Yes | Default | 0.3144 | 0.6576 | Lognormal (2) |
| 4 | Res. Water Cont. | Yes | Default | 0.1 | 0.01 | Normal (2) |
| 5 | Sat. Water Cont. | No | Default | 0.39 | 0.07 | Normal (2) |
| 6 | V. G. Alpha | Yes | Default | 5.8 | 3.8 | Normal (2) |
| 7 | V. G. Beta | Yes | Default | 1.48 | 0.13 | Normal (2) |
| 8 | Conductivity XX | No | Correction (1) | 1.0608 | 1.3512 | Lognormal (2) |
| 9 | Conductivity YY | No | Correction (1) | 1.0608 | 1.3512 | Lognormal (2) |
| 10 | Conductivity ZZ | No | Correction (1) | 1.0608 | 1.3512 | Lognormal (2) |
| 11 | Res. Water Cont. | No | Correction (1) | 0.065 | 0.02 | Normal (2) |
| 12 | Sat. Water Cont. | No | Correction (1) | 0.41 | 0.09 | Normal (2) |
| 13 | V. G. Alpha | No | Correction (1) | 7.5 | 3.7 | Normal (2) |
| 14 | V. G. Beta | No | Correction (1) | 1.89 | 0.17 | Normal (2) |
| 15 | Conductivity XX | No | Correction (2) | 7.128 | 3.744 | Lognormal (2) |
| 16 | Conductivity YY | No | Correction (2) | 7.128 | 3.744 | Lognormal (2) |
| 17 | Conductivity ZZ | No | Correction (2) | 7.128 | 3.744 | Lognormal (2) |
| 18 | Res. Water Cont. | No | Correction (2) | 0.045 | 0.01 | Normal (2) |
| 19 | Sat. Water Cont. | No | Correction (2) | 0.43 | 0.06 | Normal (2) |
| 20 | V. G. Alpha | No | Correction (2) | 14.5 | 2.9 | Normal (2) |
| 21 | V. G. Beta | No | Correction (2) | 2.68 | 0.29 | Normal (2) |

   On *Latin Hypercube Sampling|Correlation Matrix* tab, set the following parameters :

| | **3** | **4** | **6** | **7** |
|---|---|---|---|---|
| **3** | | | | |
| **4** | 0.261 | | | |
| **6** | 0.952 | 0.392 | | |
| **7** | 0.909 | -0.113 | 0.787 | |

Then click **OK**.

3. Save the project (e.g. to *3D_pump_LHS.mmb*).
4. Export and run FEMWATER-LHS.
5. Uncertainty and sensitivity analysis of the input and the output variables.

### Seawater Intrusion in Confined Aquifer

1. Start Argus ONE. In the *FEMWATER-LHS Type of Simulation Problem* window, select a **CROSS-SECTIONAL** orientation. Then click **Continue**.
2. In the FEMWATER-LHS dialog, a number of changes to the initial default values are required.

On *Model Title and Type* tab, set **NPROB =** 7

On *Time Control* tab, set:

**NTI** = 15
**TMAX** = 5000.75
**DELT** = 5
**CHNG** = 1.17169
**DELMAX** = 500

On *Fluid Properties* tab, set the coefficients for computing density and viscosity:

| coeff. A1 | Coeff. A2 | coeff. A3 | coeff. A4 | coeff. A5 | coeff. A6 | coeff. A7 | coeff. A8 |
|---|---|---|---|---|---|---|---|
| 1.0 | 0.0245 | 0.0 | 0.0 | 1.0 | 0.0 | 0.0 | 0.0 |

On *Material and Soil Properties* tab, click *Material type correction* check box to add the correction material. Set the following parameters:

On *Cond/Perm* tab, set

| xx | yy | zz | xy | xz | yz |
|---|---|---|---|---|---|
| 1.0 | 0.0 | 1.0 | 0.0 | 0.0 | 0.0 |

On *Soil Prop* tab, set

| Res MC | Sat MC | P Head | VG Alpha | VG Beta |
|---|---|---|---|---|
| 0.1 | 0.35 | 0.0 | 5.8 | 1.48 |

On *Disp/Diff* tab, set

| Dist coeff. | Bulk density | Long disper. | Trans disper. | Mol diff coeff. | Turtuosity | Decay const. | Fr N/Lang SMAX |
|---|---|---|---|---|---|---|---|
| 0.0 | 1200 | 0.0 | 0.0 | 0.066. | 1.0 | 0.0 | 0.0 |

On *Correction Cond/Perm* tab, set

| N | xx | yy | zz | xy | xz | yz |
|---|---|---|---|---|---|---|
| 1 | 0.5 | 0.0 | 0.5 | 0.0 | 0.0 | 0.0 |

On *Correction Soil Prop* tab, set

| N | Res MC | Sat MC | P Head | VG Alpha | VG Beta |
|---|---|---|---|---|---|
| 1 | 0.1 | 0.22 | 0.0 | 5.8 | 1.48 |

On *Correction Disp/Diff* tab, set

| N | Dist coeff. | Bulk density | Long disper. | Trans disper. | Mol diff coeff. | Turtuosity | Decay const. | Fr N/Lang SMAX |
|---|---|---|---|---|---|---|---|---|
| 1 | 0.0 | 1200 | 0.0 | 0.0 | 0.033. | 1.0 | 0.0 | 0.0 |

On the *Boundary Conditions|Dirichlet|Fixed Head* tab, click *Fixed Head* check box and then click **Add Rows** button to add the number of profile. Set the profiles as follows:

| N | Time 1 | Head 1 | Time 2 | Head 2 |
|---|---|---|---|---|
| 1 | 0.0 | 100.49 | 1.0e38 | 100.49 |
| 2 | 0.0 | 100.735 | 1.0e38 | 100.735 |
| 3 | 0.0 | 100.98 | 1.0e38 | 100.98 |
| 4 | 0.0 | 101.225 | 1.0e38 | 101.225 |
| 5 | 0.0 | 101.47 | 1.0e38 | 101.47 |
| 6 | 0.0 | 101.715 | 1.0e38 | 101.715 |
| 7 | 0.0 | 101.96 | 1.0e38 | 101.96 |
| 8 | 0.0 | 102.205 | 1.0e38 | 102.205 |
| 9 | 0.0 | 102.45 | 1.0e38 | 102.45 |

On the *Boundary Conditions|Dirichlet|Precribed-Concentration* tab, click *Prescr-Concentration* and set the profile:

| N | Time 1 | Conctr 1 | Time 2 | Conctr 2 |
|---|---|---|---|---|
| 1 | 0.0 | 1.0 | 1.0e38 | 1.0 |

On the *Boundary Conditions|Cauchy|Specified-Flux* tab, click *Specified-Flux* and set the profile:

| N | Time 1 | S-flux 1 | Time 2 | S-flux 2 |
|---|---|---|---|---|
| 1 | 0.0 | -6.6e-3 | 1.0e38 | -6.6e-3 |

On the *Boundary Conditions|Cauchy|Concentration* tab, click *Concentration* and set the profile:

| N | Time 1 | Conctr 1 | Time 2 | Conctr 2 |
|---|---|---|---|---|
| 1 | 0.0 | 0.0 | 1.0e38 | 0.0 |

On the *Boundary Conditions|Variable Composite|Rainfall/Evap.-Seepage* tab, click *Rainfall/Evap.-Seepage* and set the profile:

| N | Time 1 | Rf/Evap 1 | Time 2 | Rf/Evap 2 |
|---|---|---|---|---|
| 1 | 0.0 | 0.0 | 1.0e38 | 0.0 |

On the Boundary *Conditions|Variable Composite|Rainfall/Evap.*-Seepage tab, click Concentration and set the profile:

| N | **Time 1** | **Conctr 1** | **Time 2** | **Conctr 2** |
|---|---|---|---|---|
| 1 | 0.0 | 0.0 | 1.0e38 | 0.0 |

Click **OK**.

3. Activate the ***FEMWATER Quadmesh*** and import *seawater_intr_mesh.exp* into the project by selecting **File|Import FEMWATER QuadMesh...|Text File**. In CD-ROM, the *seawater_intr_mesh.exp* file is located in a directory with the pathname examples\application_4\app4_import_files\
4. Activate the ***Fixed Head Prof Type Layer1*** and import *Fixed Head Prof Type Layer1.exp* into the project by selecting **File|Import Fixed Head Prof Type Layer1...|Text File**.
5. Copy this object into ***Fixed Head Prof Type Layer2***.
6. Activate the ***Material Type Correction Layer1*** and import *Material Type Correction Layer1.exp* into the project by selecting **File|Import Material Type Correction Layer1...|Text File.**
7. Activate the ***Seepage Prof Type Layer1*** and import *Fixed Head Prof Type Layer1.exp* into the project by selecting **File|Import *Seepage* Prof Type Layer1...|Text File**.
8. Copy this object into ***Seepage Prof Type Layer2***.
9. Specify a constant slice, or default nodal slice using the **Layers dialog**. In this case, enter 0 and 1 in the **Expression dialog** for **Nodal Slice Layer1** and **Nodal Slice Layer2** respectively.
10. Save the project (e.g. to *seawater_intr.mmb*).
11. Export and run FEMWATER-LHS.
12. Using **FEMWATER Post Processing** in **PIEs** menu, plot concentration at day 4943.4 for *Contour Map* type (Fig. B.11).

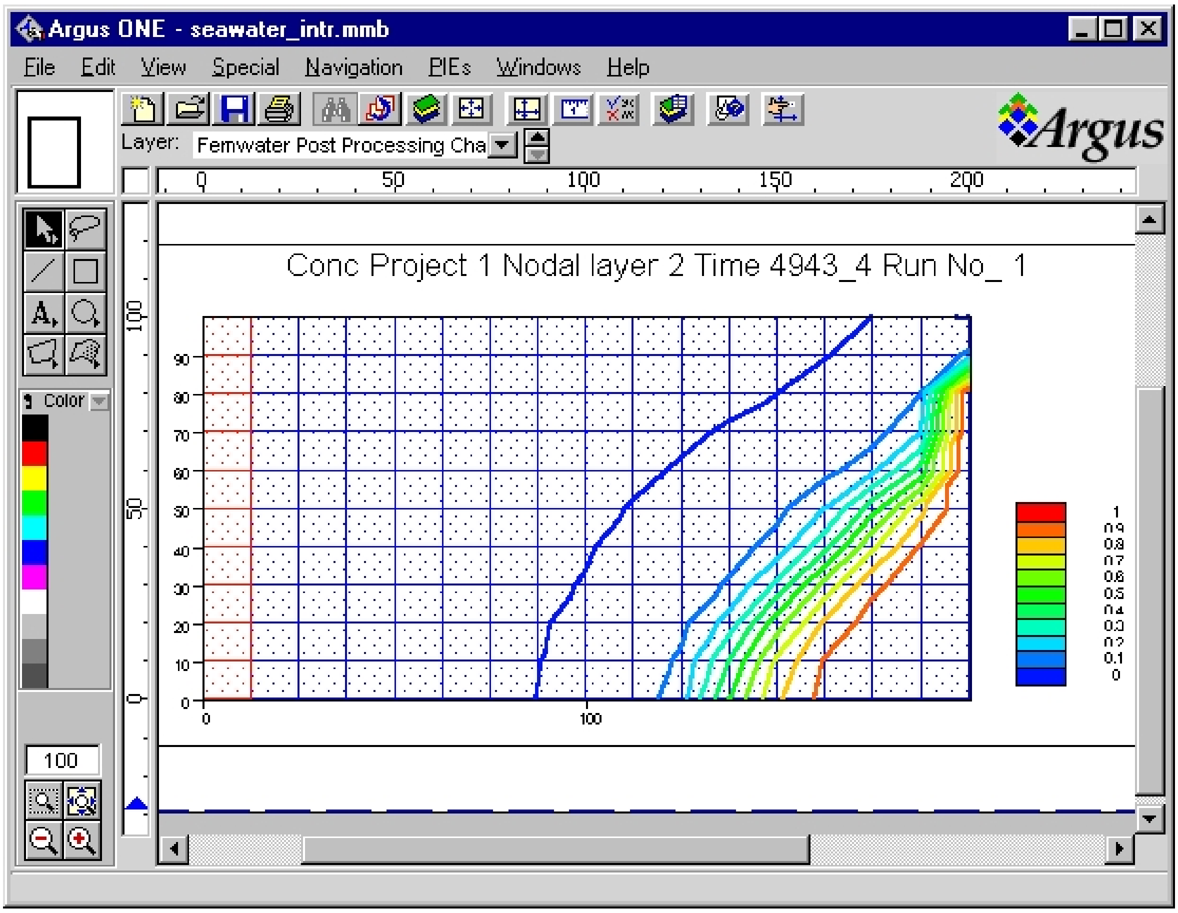


Fig. B.11 The concentration contours at the simulation time of 4943.4 days.

### Seawater Intrusion in Confined Aquifer (LHS Mode)

1. Start Argus ONE. Open the ".mmb" project file that was saved in the above case (e.g. *seawater_intr.mmb*
2. Select **PIEs**, then **Edit Project Info ...** to modify the FEMWATER-LHS Non-Spatial Information.

   On *Model Title and Type* tab, set **NPROB** = 8 and click *Latin Hypercube Sampling Simulation* check box

   On *Latin Hypercube Sampling* tab, set the *Number of Runs* to 20

   On *Latin Hypercube Sampling|Variables* tab, set the following parameters:

| N | **Variable Name** | **Correlated** | **Mat. Type** | **Min** | **Max** | **Distribution Type** |
|---|---|---|---|---|---|---|
| 1 | Conductivity XX | No | Default | 0.8 | 1.1 | Lognormal (1) |
| 2 | Conductivity ZZ | No | Default | 0.8 | 1.1 | Lognormal (1) |
| 3 | Sat. Water Cont. | No | Default | 0.315 | 0.385 | Normal (1) |
| 4 | Bulk density | No | Default | 1080 | 1320 | Normal (1) |
| 5 | Mol diff coeff. | No | Default | 0.0594 | 0.0726 | Normal (1) |
| 6 | Conductivity XX | No | Correction (1) | 0.45 | 0.55 | Lognormal (1) |
| 7 | Conductivity ZZ | No | Correction (1) | 0.45 | 0.55 | Lognormal (1) |
| 8 | Sat. Water Cont. | No | Correction (1) | 0.198 | 0.22 | Normal (1) |
| 9 | Bulk density | No | Correction (2) | 1080 | 1320 | Normal (1) |
| 10 | Mol diff coeff. | No | Correction (2) | 0.0297 | 0.0363 | Normal (1) |

Then click **OK**.

3. Save the project (e.g. to *seawater_intr _LHS.mmb*).
4. Export and run FEMWATER-LHS.
5. Uncertainty and sensitivity analysis of the input and the output variables.