2D Example for **SUTRA**

Henry2D

(Clifford I. Voss, USGS)

This 2D example is presented in the SUTRA documentation (Voss and Provost, 2002, section 6.5, page 153) and deals with simple modeling of seawater intrusion. The example reproduces a classic simulation benchmark, the Henry (1964) problem, for testing variable-density transport codes. It involves a transient saturated variable-density flow and transient solute transport simulation.

This description generally follows the discussion in the **SUTRA** documentation where figures showing the mesh and results may also be found. Step-by-step instructions for creating this example using **SutraGUI** are provided at the end of this description.

The example involves seawater intrusion into a confined aquifer modeled in cross section. Freshwater recharge inland flows into the left side of the cross section, flows over seawater in the section and discharges at a vertical sea boundary held at hydrostatic seawater pressure. The desired solution is the steady-state distribution of pressure and concentration.

The intrusion problem is nonlinear and may be solved by approaching the steady state gradually with a series of time steps. Initially there is no saltwater in the aquifer, and at time zero, seawater begins to intrude the freshwater system by moving under the freshwater from the sea boundary. The greater density of the saltwater causes the intrusion. Henry's solution assumes that dispersion is represented by a constant large coefficient of diffusion, rather than by velocity-dependent dispersivity.

Dimensions of the problem are selected to make for simple comparison with the steady-state dimensionless solution of Henry (1964). A total simulation time of t=100.0 [min], is selected, which is sufficient time for the seawater to intrude and to essentially reach a steady state position at the scale simulated.

The main parameter values are:

$\varepsilon = 0.35$

$k = 1.020408 \times 10^{-9} \ [\text{m}^2]$
(based on $K=1.0 \times 10^{-2} \ [\text{m/s}]$)

$C_{\text{sea}} = 0.0357 \left[\dfrac{\text{kg(dissolved solids)}}{\text{kg(seawater)}} \right]$

$B = 1.0 \ [\text{m}]$

$\rho_{\text{sea}} = 1024.99 \ [\text{kg/m}^3]$

$\alpha_L = \alpha_T = 0.0 \ [\text{m}]$

$\dfrac{\partial \rho}{\partial C} = 700. \left[\dfrac{\text{kg(seawater)}^2}{\text{kg(dissolved solids) m}^3} \right]$

$|\underline{g}| = 9.8 \ [\text{m/s}^2]$

$\rho_o = 1000. \ [\text{kg/m}^2]$
$D_m = 18.8571 \times 10^{-6} \ [\text{m}^2/\text{s}]$
$Q_{\text{IN}} = 6.6 \times 10^{-2} \ [\text{kg/s}]$

$C_{\text{IN}} = 0.0 \left[\dfrac{\text{kg(dissolved solids)}}{\text{kg(water)}} \right]$

Boundary Conditions:

A freshwater source is set along the left vertical boundary and is implemented by employing source nodes along the left vertical boundary, which inject freshwater at a rate of Q_{IN}, and concentration of C_{IN}. Specified pressure is set at hydrostatic seawater pressure with ($_{\text{sea}}$=1024.99 [kg/m^3]) along the right vertical boundary through the use of specified pressure nodes. Any inflowing fluid at this boundary has the concentration, C_{sea}=0.0357 [kg(dissolved solids)/kg(seawater)], of seawater. No flow occurs across the top and bottom boundaries.

Initial Conditions:

Freshwater concentration and natural steady-state pressures are initially set everywhere in the aquifer. The natural initial pressure values are obtained through an extra initial simulation that calculates steady pressures for the conditions of freshwater concentration (C=0) throughout, with the boundary conditions described above.

Mesh:

The mesh consists of twenty by ten elements, each of size 0.1 [m] by 0.1 [m], (NN=231, NE=200). Mesh thickness, B, is 1.0 [m].

Simulation:

A natural steady-state pressure distribution is obtained in a pre-simulation (described above under *Initial Conditions*).

For the transient runs, for both pressure and concentration, the time step size is $\Delta t = 60.$ s. Because only the long-time (steady-state) behavior of the system is of interest, a single iteration for resolving nonlinearities is used per time step. The system essentially achieves a new steady state after 100 time steps (100 min). For 2D, the direct solver is used for both simulations. Both pressure and concentration are solved for on each time step (NUCYC=NPCYC=1).

The transient run takes about 1 second on a PC running Windows 2000 with a 2GHz Xeon processor; it uses less than 1 Mbyte of RAM.

SUTRA *files:*

For initial run to obtain starting pressures
> input :
> *Henry2D_initial-p.inp*, *Henry2D_initial-p.ics*, and SUTRA.FIL
>
> output:
> *Henry2D_initial-p.smy*, *Henry2D_initial-p.lst*, *Henry2D_initial-p.rst*, *Henry2D_initial-p.nod*, and *Henry2D_initial-p.ele*.

For transient run
> input :
> *Henry2D.inp*, *Henry2D.ics*, and SUTRA.FIL
>
> output:
> *Henry2D.smy*, *Henry2D.lst*, *Henry2D.rst*, *Henry2D.nod*, *Henry2D.ele*, and *Henry2D.obs*.

Preprocessing:

The ArgusONE setup files, *Henry2D_initial-p.mmb* and *Henry2D.mmb*, are included for use with **SutraGUI**. In **SutraGUI**, this problem uses a fishnet mesh with eight blocks. The top of the model represents the top of the island on land and the sea bottom below the sea. The top is assigned an elevation, given as a function of radius from the center of the island. The bottom of the upper block is 5. m below the top and the bottom of the model is at 100 m depth below sea level. Recharge is specified on the land portion of the island on the top surface of the model with zero concentration of inflow. It is adjusted with radius as described above. On the sea bottom, the pressure is specified as hydrostatic

seawater. The outermost vertical side of the model is assigned a specified pressure of hydrostatic seawater.

A line of observation points is placed along the bottom of the cross section near the sea for the transient simulation.

In order to begin the transient simulation with a natural pressure distribution, an initial steady-state simulation is done (using the ArgusONE setup file *Henry2D_initial-p.mmb*) to determine the pressures in the model prior to intrusion (when all ground water is freshwater). The pressures resulting from this simulation are used as the initial pressure condition for the transient simulation (that uses using the ArgusONE setup file *Henry2D.mmb*). This is specified below "Select restart file:" on the <u>Initial Conditions Controls</u> page of the <u>SUTRA Project Information</u> dialog.

> *Note:* The user must change the path of the restart file so that it is correct on the user's computer).

Results:

Results are reported 100 min after intrusion begins, by which time the system has nearly reached a new steady state. (Note that more-exact steady-state solutions may be obtained by running longer simulations, e.g. 1000 min.)

Postprocessing:

The match of specified and calculated pressures may be checked using **CheckMatchBC**.

Hydrographs of pressure and concentration at the observation nodes may be viewed using **GW_Chart**.

Spatial distributions of results may be viewed with **ArgusONE**, **SutraPlot** and **ModelViewer**.

> For **ArgusONE**, the plot shows contours of concentration (10% to 90% seawater in 10% increments) and flow vectors.

> For **SutraPlot**, two setup files are provided; *Henry2D_mesh.spl*, shows the mesh and, *Henry2D_conc.spl*, shows contours of percent seawater (between 10% and 90%) in a vertical section at the end of the simulation.

> For **ModelViewer**, a setup file, *Henry2D.mv*, is provided for viewing the results with isosurfaces (contours) for concentration (10% to 90% seawater) and velocity vectors. The first image shows the initial

conditions. To see the end of the simulation, select Toolbox|Animation|Set to time: (select 100)|Set. **ModelViewer** can only visualize results from FishNet meshes in 2D, not from irregular meshes.

Reference:

Henry, H.R., 1964, Effects of dispersion on salt encroachment in coastal aquifers: in Sea Water in Coastal Aquifers: U.S. Geological Survey Water-Supply Paper 1613-C, p. C71-C84.

Step-by-Step Instructions
for Creating the **Henry** Example
Using **SutraGUI** and **SUTRA**

This exercise deals with simple modeling of seawater intrusion by setting up and running the original Henry (1964) problem. The exercise reproduces a classic simulation benchmark, the Henry (1964) problem, for testing variable-density transport codes. This benchmark is presented as an example simulation in the **SUTRA** documentation (Voss and Provost (2002), section 6.5, page 153).

The simulation is set up using the **SutraGUI** graphical interface. It involves creating a perfectly rectangular cross-sectional model domain containing a Fishnet Mesh, and applying boundary conditions exactly along the two vertical edges of the domain. Along one edge, a total fluid inflow is specified and **SutraGUI** is allowed to distribute the value of fluid sources along the edge. Along the other edge, you will create a specified pressure condition representing hydrostatic seawater in a single object along which pressure linearly increases with depth.

Step-by-Step Instructions

----

1. Start *ArgusONE*.

   PIEs → New SUTRA Project…
   (found along the top of the window)
   In the *SUTRA Project Information* window that appears, on the
   Configuration tab, select:
   - 2 Dimensions
   - Cross-sectional orientation
   - Solute transport with variable-density fluid, using pressure
   - User-specified model thickness
   - Fishnet mesh

   Then click OK.

2. Make the *ArgusONE* window that appears full screen. Also, resize the
   Layers window that appears so that you can see all of the layer names.

3. Then, to set work area size, select **Special → Drawing Size…** and set
   size as follows:

   Horizontal Extent: =   3
   Vertical Extent: =   2
   Horizontal Origin: =   -0.5
   Vertical Origin: =   -0.5

   Click OK.

4. The workspace shrinks because of the reduced drawing size. Resize it
   using the *Zoom to Fit* tool in the lower left corner of the window.

The next steps create the 2 x 1 model domain for the Henry simulation by first
approximating the shape by hand drawing, and then correcting it.

5. Activate the **Fishnet_Mesh_Layout layer**.

6. Click on the quadrilateral element drawing tool near the upper left edge of the window.

7. While holding down the shift key on the keyboard, draw the rectangle representing the model domain by placing four vertices at about (0,0), (0,1), (2,1) and (2,0). (Holding down the shift key while drawing causes lines to be drawn horizontally and vertically.)

8. Select the Navigate arrow at the upper left edge of the window and double-click on the block just drawn. This brings up the *Element Information dialog*. Set:

   elements_in_x = 20
   elements_in_y = 10

   This information is used later when the finite element mesh is created.

   Click **OK**.

9. Click outside of the block so that it becomes unselected (not black). Double-click on each of the four vertices, in turn. In the *Node Information dialog* that appears, correct the vertex coordinates to make them exact (e.g. x=0.0, y=0.0), clicking **OK** after each correction is complete.

10. Save the project by selecting **File → Save**.

The next steps create a line source exactly along the left vertical edge of the domain representing inflow of fresh water to the cross section. The total inflow across this boundary (0.066) and the solute concentration of inflowing fresh-water (0.0) are specified.

11. To copy the left edge from the *Fishnet_Mesh_Layout* to the *Sources of Fluid* layer, click on **PIEs → Convert... → Mesh Objects To Contours...**.
   In the dialog that appears, first select *Fishnet_Mesh_Layout*, **OK**.
   Then select *Sources of Fluid* , **OK**.
   The *Mesh Objects To Contours window* appears.

12. Click on the left vertical side of the block. This highlights the left edge and vertices. Click **OK** (then answer No to the question about 'intersection') and this line is pasted into the *Sources of Fluid layer*.

13. Activate the *Sources of Fluid layer* to see the pasted line.

14. Double click on the open contour just pasted (i.e., the copied line) to bring up its *Contour Information dialog*.

15. The parameter values must be specified for this object. The user has a choice of specifying either a total source value for the open contour object, or a source per length of the object. Only one of these may have a value, and the other **must** be set to **$N/A**, meaning 'undefined'. This object represents the boundary condition for inflowing fresh water in the cross section.
    Set:

total_source	=	0.066
specific_source	=	$N/A
concentration_of_source	=	0.0

    and click **OK**.

    Save the project.

The following steps create a background value for the **Specified Pressure layer** that represents the pressure of seawater under hydrostatic conditions. This background value provides a pressure everywhere in the workspace. Sea level is set at elevation y=1 (i.e., the top of the model) and pressure increases below this point.

The density of seawater is given by the linear fluid density expression used by SUTRA, (1000. + 700. * 0.0357), gravity is 9.8 m/s^2, and depth is given by (1. - y).

16. Open the *Layers dialog* by clicking the **Layers...** button in the *Layer List* window.

17. In the upper *Layers* window, click on the **Specified Pressure layer** to select it (it then becomes black).

18. In the lower *Layer Parameters:* window, click on the f_x button in the **Value** column of the row labeled **specified_pressure**.

19. In the *Expression dialog* that appears, type the following equation:

    (1000. + 700. * 0.0357) * 9.8 * (1. - Y())

(The Y() is an ArgusONE function and may be automatically inserted by clicking on **Mathematical** (under **Function**s) in the lower left list, and then clicking on **Y** in the lower right list.)

Then click **OK**.

20. Select the **concentration** row in the **Layer Parameters** table (lower window),
Then click on f_x in the **Value** column.

21. In the *Expression dialog*, set the value to 0.0357, and click **OK**.

Click **Done** in the Layers *dialog*.

Save the project.

22. Activate the ***Specified Pressure layer*** and move the cursor up and down in the workspace to observe the varying background pressure value just specified. (The value is shown in the lower left margin of the window.) Note that zero pressure occurs along the top edge of the model domain and pressure increases with depth.

The next steps create a specified pressure boundary condition exactly along the right vertical edge of the model domain by copying the right edge from the ***Fishnet_Mesh_Layout layer*** to the ***Specified Pressure layer***. When this model domain is meshed, nodes falling exactly along this line will be automatically assigned both specified pressure values varying with depth and seawater concentration to create the desired boundary condition. The line receives values from the background pressures set just above.

23. To copy the right edge from the ***Fishnet_Mesh_Layout*** to the ***Specified Pressure layer***, click on **PIEs → Convert → Mesh Objects To Contours…**.
In the dialog that appears, first select ***Fishnet_Mesh_Layout***, **OK**.
Then select ***Specified Pressure***, **OK**.
The *Mesh Objects To Contours window* appears.

24. Click on the right vertical side of the block. This highlights the right edge and vertices. Click **OK** (then answer No to the question about 'intersection') and this line is pasted into the ***Specified Pressure layer***.

25. Activate the ***Specified Pressure layer*** to see the pasted line.
Save the project.

This completes entry of spatially-varying information for the Henry problem.

In the next steps, the constant parameter values for various layers will be set to the values required for the Henry problem. This can be done while adding non-spatial information to the project.

26. **PIEs → Edit Project Info...**

In the *SUTRA Project Information dialog* that appears, a number of changes to the initial default values are required to match the Henry problem.

Click on the **Parameter Values – Quick Set** button at the bottom of the dialog and the *Parameter Values – Quick Set dialog* appears:
   set **Maximum Permeability** to 1.020408E-9, click *Set Now*
   set **Minimum Permeability** to 1.020408E-9, click *Set Now*
   set *both* **Longitudinal Dispersivities** to 0, click *Set Now*
   set *both* **Transverse Dispersivities** to 0, click *Set Now*
   set **Porosity** to 0.35, click *Set Now*
Click **Done** to close this dialog.

Headings pane:
   set **TITLE1** to "Henry Problem from SUTRA Documentation"
   set **TITLE2** to "variable-density benchmark problem"

Modes pane:
   select **Transient ground-water flow. Transient solute transport**

Temporal Controls pane:
   set **ITMAX** = 100 (number of time steps)
   set **DELT** = 60. (time step size in s)
   set **TMAX** = 1.E9 (maximum simulation time)

Output Controls pane:
   set **NPRINT** = 50 (print cycle for main listing)
   set **NCOLPR** = 50 (print cycle for node results)
   set **LCOLPR** = 50 (print cycle for velocities)

Fluid Properties pane:
   set **SIGMAW** = 18.8571e-6
   set **VISC0** = 1.e-3 (default value)

Production, Gravity pane:

set **GRAVY** = -9.80

Then click **OK**.

Save the project.

Next, the mesh is created.

27. Activate the ***SUTRA Mesh layer***.

28. To create the mesh, select **PIEs → Create SUTRA Fishnet Mesh**. This creates the Fishnet mesh that now appears on the screen. (Compare this with the mesh shown on page 197 of the SUTRA documentation.)

29. Inspect the values assigned to nodes and elements in the mesh by double clicking the node or element of interest.

Check the distribution of inflow (**QIN**) along the left-hand boundary nodes, where the upper and lower corner nodes are assigned half the inflow of the others along the boundary to correctly create a uniform inflow. The concentration of inflowing fluid (**UIN**) should be zero.

Note also that the specified pressure (**PBC**) increases with depth at the right-hand boundary nodes, with values equivalent to the pressure of a hydrostatic column of seawater. The concentration of any inflowing fluid at these nodes (**pUBC**) should be that of seawater, 0.0357.

30. Save the project.

Finally, run the simulation, and plot the results.

31. **PIEs → Run SUTRA**. In the *Run SUTRA window* that appears, type in a root file name, "Henry2D". Then click **OK**.

32. In the *Save As window* that appears, select the directory into which the Sutra input and output files will be written. Then click **Save**. The SutraGUI exports the Sutra input files and then the Sutra simulation runs in the black window that appears. Close the black window by typing any key.

33. Plot concentration and velocity results for time step 100.
Select **PIEs → SUTRA 2D Post Proc...**.

Check that the file, Henry2D.nod, is selected, and click **Open**.
The *Select SUTRA results to visualize dialog* appears.
First select Concentration and Velocity for Time Step 100 by clicking on the appropriate "no" until "YES" appears.
Then click **OK**.
To see the plot, activate the ***SUTRA Post Processing Charts layer***.

34. Then, to plot each 10% of seawater concentration, double-click on or to the right of the color bar, and in the *Contour Diagram dialog* that appears:

set **Minimum:**   =   0.00357
set **Maximum:**   =   0.0357
set **Delta:**   =   0.00357

Click **OK**.

35. Then, in order to compare with the results found in the SUTRA documentation on page 201, plot each 25% by setting both **Minimum:** and **Delta:** to 0.008925.

Save the project.

* * * * * *