Model created in COMSOL Multiphysics 6.4

Seawater Intrusion in a Coastal Aquifer

Introduction

Seawater intrusion is a significant concern in coastal regions, where the balance between freshwater sources and saltwater infiltration is constantly under threat. This problem is complex because it not only threatens the supply of fresh water but also the quality of groundwater.

This example illustrates how to set up a model for saltwater intrusion in a coastal aquifer when a pumping well is positioned at some distance from the shoreline. The inspiration for this approach arose from the research paper titled “Preferential Flow Enhances Pumping-Induced Saltwater Intrusion in Volcanic Aquifers.” In this research, a range of methods were utilized to replicate the process of saltwater intrusion in a volcanic aquifer distinguished by the presence of highly conductive “lava tubes”, contrasted with other geological formations displaying comparatively lower conductivity levels.

Given the distinctive characteristics of this scenario, where a low-conductivity aquifer intersects with highly conductive tubes, our model incorporates both the homogenized porosity approach as well as the dual porosity approach to capture these conditions.

Model Definition

Figure 1 illustrates the model’s geometry and scenario, with the sea level represented by a hydraulic head condition. The model assumes a consistent rate of freshwater recharge, and the land surface is subjected to precipitation.

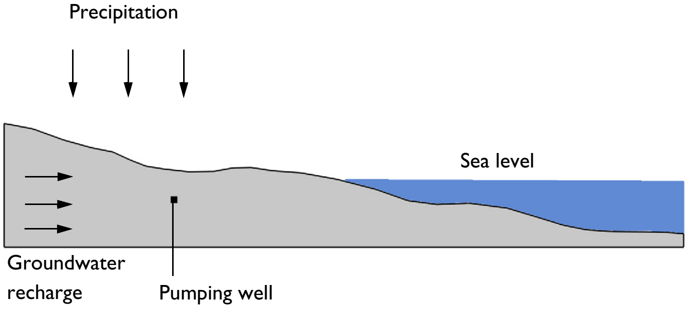

Figure 1: Model geometry and conditions.

Freshwater has a density of $\rho_f = 1000\ \text{kg/m}^3$, while saltwater has a higher density of $\rho_s = 1025\text{kg/m}^3$. These density differences are essential and are incorporated into the model by considering the transport of salt within the aquifer and utilizing a linearized density relationship:

$$\rho = \rho_f + (\rho_s - \rho_f)\frac{c}{c_s} \tag{1}$$

Saltwater typically has a salinity of approximately 35% or 35 g/l. In this model, the specific value of c_s (maximum salt concentration) is not critical, as all effects are contingent upon the relative salt concentration c. Nonetheless, realistic salt concentration values are utilized by assuming a molar mass of 58.44 g/mol, resulting in $c_s = 598.9 \mathrm{mol/m}^3$.

The transport through the porous aquifer is governed by the conservative formulation of the diffusion–convection equation which also includes dispersion.

$$\varepsilon_p \frac{\partial c}{\partial t} + \nabla \cdot (-(D_d + D_e)\nabla c + \mathbf{u}c) = Q_s \tag{2}$$

here, D_d and D_e are the dispersion and diffusion coefficients., ε_p is the porosity , and on the right hand side Q_s denotes the source term. The convective velocity $\mathbf{u}$ is calculated using Darcy's law:

$$\mathbf{u} = -\frac{K}{\rho g}(\nabla p - \rho \mathbf{g}) \tag{3}$$

together with the continuity equation

$$\frac{\partial}{\partial t}(\varepsilon_p \rho) + \nabla \cdot (\rho \mathbf{u}) = Q_m \tag{4}$$

Here, K denotes the hydraulic conductivity, g and $\mathbf{g}$ are the gravity constant and vector, respectively, while Q_m represents a source term. Note that the density ρ is given by Equation 1.

In Ref. 1 the researchers conducted a comparison of different approaches, including a homogenized approach and various heterogeneous approaches to represent diverse facies with distinct conductivities. They specifically set the conductivity for lava tubes to be two orders of magnitude larger than the next highest facies, sparking the idea of investigating a dual porosity approach.

In this example, a homogenized approach is initially considered assuming an anisotropic hydraulic conductivity where horizontal conductivity exceeds vertical conductivity. Then the solution is compared to that of a dual porosity approach.

Dual porosity implies that the flow occurs within the highly conductive area only, namely the lava tubes, which constitute the macroporous part of the system. In contrast, flow within all other facies is stagnant due to their significantly lower conductivity, forming the microporous system. However, these facies can still exchange fluids with the tubes, acting as sources or sinks. Consequently, Equation 3 and Equation 4 are employed for the volume fraction of the macropores (θ_M). The source term Q_m represents the interporosity flow and depends on the pressure difference between the macro- and micropores

$$Q_m = -\alpha_w(p_M - p_m) \tag{5}$$

The fluid transfer function α_w (s/m^2)is influenced by various factors such as the facies structure and characteristics, and it is typically not known with accuracy. A more detailed discussion can be found in Ref. 2. Smaller values of α_w indicate longer fluid exchange times. Therefore, when observation periods are sufficiently long and a steady state regime is achieved, the precise value becomes less significant. For the volume fraction of the micropores only an ordinary differential equation (ODE) needs to be solved:

$$(1-\theta_M)\frac{\partial}{\partial t}(\varepsilon_p\rho) = -Q_m$$

Above equations are incorporated using the **Dual Porosity** feature within the Darcy's Law interface. To account for the dual porosity characteristics in salt transport, this is manually integrated by introducing an additional ODE for the micropores:

$$(1-\theta_M)\varepsilon_p\frac{\partial c_m}{\partial t} = -Q_s$$

in conjunction with a mass source term within the transport equation for the macropores (Equation 2) with the mass source being:

$$Q_s = -\alpha_s(c - c_m)$$

Like for the fluid transfer function α_w, the mass transfer function α_s (1/s) is not known and discussed in Ref. 2.

Results and Discussion

The model is calculated over a duration of one year. Notably, the concentration distribution remains relatively stable after approximately 100 days. The entire year is simulated to account for varying precipitation rates (Figure 2), although it demonstrates that their impact is relatively minor.

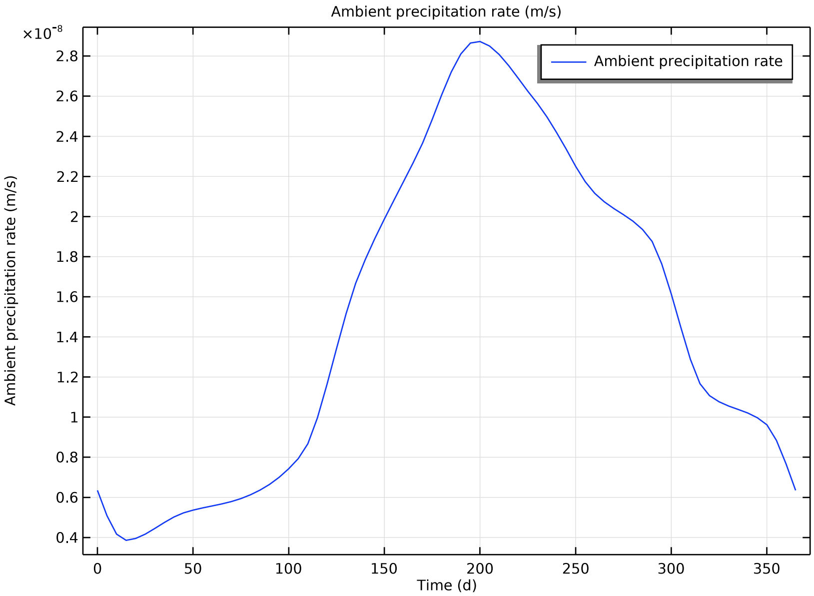

Figure 2: Varying precipitation rate over one year.

Figure 3 shows the pressure.

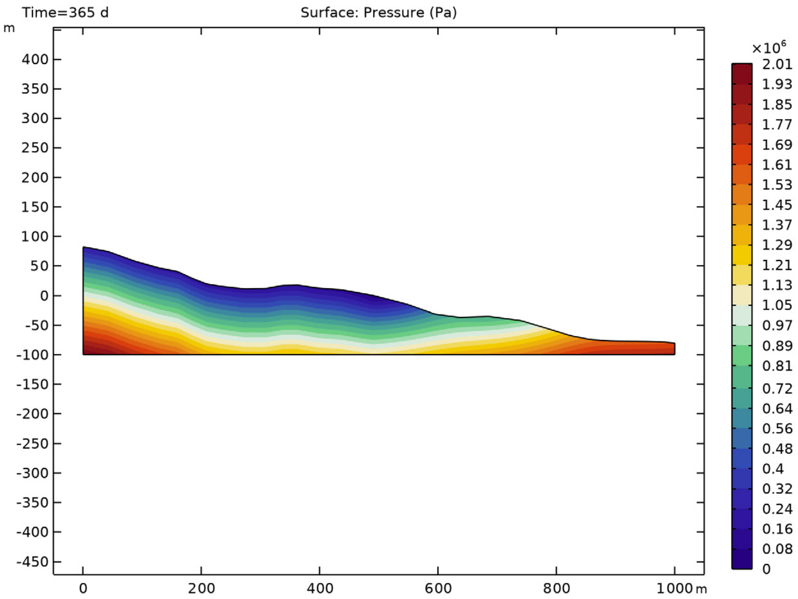

Figure 3: Pressure distribution after one year.

The concentration plot shows the salt concentration for the dual porosity approach. A contour line marks the location of the interface between saltwater and freshwater, where the concentration is 17.5 g/mol, representing half the concentration found in saltwater in the homogenized approach.

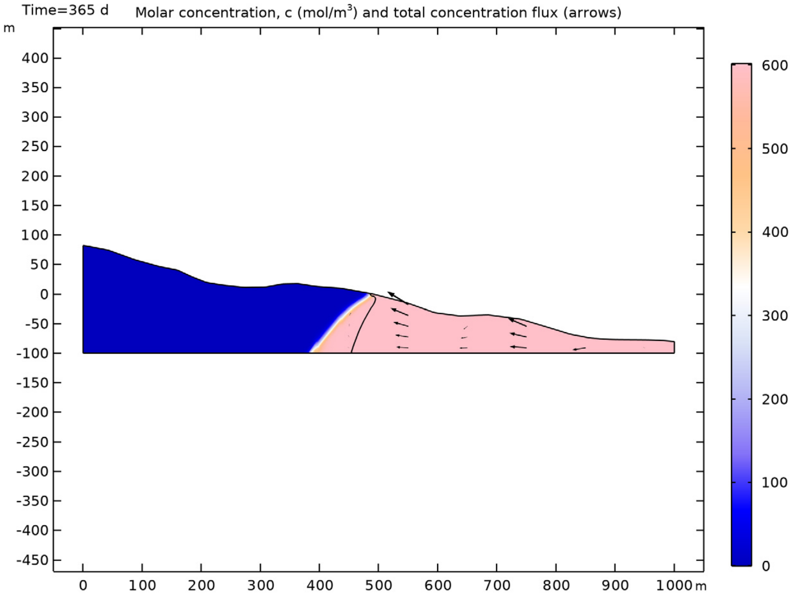

Figure 4: Concentration distribution and total concentration flux (arrows) after one year for the dual porosity approach. The black contour line indicates the position of the freshwater–saltwater interface for the homogenized approach.

The outcome of the dual-porosity simulation is that even though the volume fraction of the highly conductive lava tubes is small compared to the aquifer's low-conductive constituents, they can have a large impact on the saltwater intrusion characteristics. The results show that the dual-porosity approach predict a larger saltwater intrusion as compared to the homogenized approach.

Note that the parameters in this example are chosen arbitrarily. To predict saltwater intrusion in real aquifers it is essential to have good experimental data of the aquifer composition.

References

1. X. Geng and H.A. Michael, "Preferential flow enhances pumping-induced saltwater intrusion in volcanic aquifers," *Water Resour. Res.*, vol. 56, 2020; doi.org/10.1016/j.jhydrol.2022.127835.

2. T. Vogel, H.H. Gerke, R. Zhang, and M.Th. Van Genuchten, “Modeling flow and transport in a two-dimensional dual-permeability system with spatially variable hydraulic properties,” *J. Hydrol.*, vol. 238, nos. 1–2, pp. 78–89, 2000. doi.org/10.1016/S0022-1694(00)00327-9.

Application Library path: `Subsurface_Flow_Module/Solute_Transport/seawater_intrusion`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, click **2D**.
2. In the **Select Physics** tree, select **Fluid Flow** > **Porous Media and Subsurface Flow** > **Darcy’s Law (dl)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Chemical Species Transport** > **Transport of Diluted Species in Porous Media (tds)**.
5. Click **Add**.
6. Click **Study**.
7. In the **Select Study** tree, select **General Studies** > **Time Dependent**.
8. Click **Done**.

GLOBAL DEFINITIONS

Start by loading parameters and the geometry sequence into the model.

Parameters 1

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. Click **Load from File**.

4. Browse to the model's Application Libraries folder and double-click the file `seawater_intrusion_parameters.txt`.

GEOMETRY 1

1. In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
2. Browse to the model's Application Libraries folder and double-click the file `seawater_intrusion_geom_sequence.mph`.
3. In the **Geometry** toolbar, click **Build All**.

DEFINITIONS

Next, define a variable for the density to account for density changes due to salt concentration.

Variables 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
2. In the **Settings** window for **Variables**, locate the **Variables** section.
3. In the table, enter the following settings:

Name	Expression	Unit	Description
`rho`	`rhof+(rhos-rhof)*c/cs`	kg/m³	`Seawater density`

DARCY'S LAW (DL)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Darcy's Law (dl)**.
2. In the **Settings** window for **Darcy's Law**, locate the **Gravity Effects** section.
3. Select the **Include gravity** checkbox.

Gravity 1

The sea level is at $y = 0$m. To accurately account for the influence of gravity effects, the reference position is modified to align with the surface elevation for locations above sea level ($y > 0$) and is set to 0 for locations below sea level ($y < 0$).

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Darcy's Law (dl)** click **Gravity 1**.
2. In the **Settings** window for **Gravity**, locate the **Gravity** section.
3. Select the **Specify reference position** checkbox.

4 Specify the $\mathbf{r}_{\mathrm{ref}}$ vector as

`int1(x)*(int1(x)>0)`	`y`

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Darcy's Law (dl)** > **Porous Medium 1** click **Fluid 1**.

2 In the **Settings** window for **Fluid**, locate the **Fluid Properties** section.

3 From the ρ list, choose **User defined**. In the associated text field, type `rho`.

Porous Matrix 1

1 In the **Model Builder** window, click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the ε_{p} list, choose **User defined**. In the associated text field, type `por`.

4 From the **Permeability model** list, choose **Hydraulic conductivity**.

5 From the list, choose **Diagonal**.

6 Specify the K matrix as

`Kmean_h`	`0`
`0`	`Kmean_v`

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Darcy's Law (dl)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Click the **Hydraulic head** button.

Hydraulic Head 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Hydraulic Head**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Hydraulic Head**, locate the **Hydraulic Head** section.

4 In the H_0 text field, type `abs(y)`.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4. In the U_0 text field, type `q_in`.

Well 1

1. In the **Physics** toolbar, click **Points** and choose **Well**.
2. Select Point 3 only.
3. In the **Settings** window for **Well**, locate the **Well** section.
4. From the **Well type** list, choose **Production**.
5. From the **Specify** list, choose **Mass flow**.
6. Locate the **Mass Flow** section. In the M_0 text field, type `Mwell*dl.rho`.

DEFINITIONS

Ambient Properties 1 (ampr1)

1. In the **Physics** toolbar, click **Shared Properties** and choose **Ambient Properties**.
2. In the **Settings** window for **Ambient Properties**, locate the **Ambient Settings** section.
3. From the **Ambient data** list, choose **Meteorological data (ASHRAE 2021)**.
4. Locate the **Location** section. Click **Set Weather Station**.
5. In the **Weather Station** dialog, type `Hono` in the text field.
6. In the tree, select **Oceania** > **United States** > **HONOLULU INTL (911820)**.
7. Click **OK**.

DARCY'S LAW (DL)

Precipitation 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Precipitation**.
2. Select Boundary 4 only.
3. In the **Settings** window for **Precipitation**, locate the **Precipitation** section.
4. From the P_0 list, choose **Ambient precipitation rate (ampr1)**.

TRANSPORT OF DILUTED SPECIES IN POROUS MEDIA (TDS)

Fluid 1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Transport of Diluted Species in Porous Media (tds)** > **Porous Medium 1** click **Fluid 1**.
2. In the **Settings** window for **Fluid**, locate the **Convection** section.
3. From the $\mathbf{u}$ list, choose **Total Darcy velocity field (dl/porous1)**.
4. Locate the **Diffusion** section. In the $D_{\mathrm{F,c}}$ text field, type `D`.

Porous Matrix 1

1. In the **Model Builder** window, click **Porous Matrix 1**.
2. In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.
3. From the ε_p list, choose **User defined**. In the associated text field, type `por`.

Porous Medium 1

In the **Model Builder** window, click **Porous Medium 1**.

Dispersion 1

1. In the **Physics** toolbar, click **Attributes** and choose **Dispersion**.
2. In the **Settings** window for **Dispersion**, locate the **Dispersion** section.
3. From the **Dispersion tensor** list, choose **Dispersivity**.
4. In the α_L text field, type `4`.
5. In the α_T text field, type `0.4`.

Concentration 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Concentration**.
2. Select Boundary 5 only.
3. In the **Settings** window for **Concentration**, locate the **Concentration** section.
4. Select the **Species c** checkbox.
5. In the $c_{0,c}$ text field, type `cs`.

Concentration 2

1. In the **Physics** toolbar, click **Boundaries** and choose **Concentration**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Concentration**, locate the **Concentration** section.
4. Select the **Species c** checkbox.

MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
3. From the **Element size** list, choose **Extremely fine**.
4. Right-click **Component 1 (comp1)** > **Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** click **Size**.
2. In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

3. In the **Maximum element size** text field, type `5`.
4. Click **Build All**.

STUDY 1: HOMOGENIZED APPROACH

1. In the **Model Builder** window, click **Study 1**.
2. In the **Settings** window for **Study**, type `Study 1: Homogenized approach` in the **Label** text field.

Step 1: Time Dependent

1. In the **Model Builder** window, under **Study 1: Homogenized approach** click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3. From the **Time unit** list, choose **d**.
4. In the **Output times** text field, type `range(0,5,365)`.
5. In the **Study** toolbar, click **Compute**.

   By default, plots for the pressure, velocity, and concentration are created automatically. Before examining the results, compute the solution for the dual porosity approach.

DARCY'S LAW (DL)

Dual Porosity Medium 1

1. In the **Physics** toolbar, click **Domains** and choose **Dual Porosity Medium**.
2. Select Domain 1 only.
3. In the **Settings** window for **Dual Porosity Medium**, locate the **Interporosity Flow** section.
4. In the α_w text field, type `1e-3`.

Fluid 1

1. In the **Model Builder** window, click **Fluid 1**.
2. In the **Settings** window for **Fluid**, locate the **Fluid Properties** section.
3. From the ρ list, choose **User defined**. In the associated text field, type `rho`.
4. From the μ list, choose **User defined**.

Macropores 1

1. In the **Model Builder** window, click **Macropores 1**.
2. In the **Settings** window for **Macropores**, locate the **Matrix Properties** section.
3. From the **Permeability model** list, choose **Hydraulic conductivity**.
4. Locate the **Volume Fraction** section. In the θ_M text field, type `theta_M`.

5. Locate the **Matrix Properties** section. From the $\varepsilon_{\mathrm{p,M}}$ list, choose **User defined**. In the associated text field, type `por`.
6. From the **Permeability model** list, choose **Power law**.
7. In the b_{PL} text field, type `bPL`.
8. In the n_{PL} text field, type `nPL`.

Micropores 1

1. In the **Model Builder** window, click **Micropores 1**.
2. In the **Settings** window for **Micropores**, locate the **Matrix Properties** section.
3. From the $\varepsilon_{\mathrm{p,m}}$ list, choose **User defined**. In the associated text field, type `por`.

ADD PHYSICS

1. In the **Physics** toolbar, click **Add Physics** to open the **Add Physics** window.
2. Go to the **Add Physics** window.
3. In the tree, select **Mathematics** > **ODE and DAE Interfaces** > **Domain ODEs and DAEs (dode)**.
4. Click the **Add to Component 1** button in the window toolbar.
5. In the **Physics** toolbar, click **Add Physics** to close the **Add Physics** window.

DOMAIN ODES AND DAES (DODE)

1. In the **Settings** window for **Domain ODEs and DAEs**, locate the **Units** section.
2. Click **Select Dependent Variable Quantity**.
3. In the **Physical Quantity** dialog, type `conc` in the text field.
4. In the tree, select **General** > **Concentration (mol/m^3)**.
5. Click **OK**.
6. In the **Settings** window for **Domain ODEs and DAEs**, locate the **Units** section.
7. Click **Select Source Term Quantity**.
8. In the **Physical Quantity** dialog, type `reac` in the text field.
9. In the tree, select **Transport** > **Reaction rate (mol/(m^3*s))**.
10. Click **OK**.
11. In the **Settings** window for **Domain ODEs and DAEs**, click to expand the **Dependent Variables** section.
12. In the **Field name (mol/m^3)** text field, type `c_m`.

13. In the **Dependent variables (mol/m³)** table, enter the following settings:

`c_m`

Distributed ODE 1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Domain ODEs and DAEs (dode)** click **Distributed ODE 1**.
2. In the **Settings** window for **Distributed ODE**, locate the **Damping or Mass Coefficient** section.
3. In the d_a text field, type `por`.
4. Click to expand the **Equation** section. Locate the **Source Term** section. In the f text field, type `-Qs/(1-theta_M)`.

DEFINITIONS

Variables 1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Definitions** click **Variables 1**.
2. In the **Settings** window for **Variables**, locate the **Variables** section.
3. In the table, enter the following settings:

Name	Expression	Unit	Description
`Qs`	`-1e-4[1/s]*(c-c_m)`	mol/(m³·s)	`Mass exchange term`

TRANSPORT OF DILUTED SPECIES IN POROUS MEDIA (TDS)

In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species in Porous Media (tds)**.

Heterogeneous Reactions 1

1. In the **Physics** toolbar, click **Domains** and choose **Heterogeneous Reactions**.
2. Select Domain 1 only.
3. In the **Settings** window for **Heterogeneous Reactions**, locate the **Reaction Rates** section.
4. In the R_c text field, type `Qs/theta_M`.
5. Locate the **Reacting Volume** section. From the list, choose **Total volume**.

ADD STUDY

1. In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Time Dependent**.

4 Click the **Add Study** button in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 From the **Time unit** list, choose **d**.

3 In the **Output times** text field, type `range(0,5,365)`.

4 In the **Model Builder** window, click **Study 2**.

5 In the **Settings** window for **Study**, type `Study 2: Dual porosity approach` in the **Label** text field.

6 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox, because the plots that have already been generated will be utilized again.

7 In the **Study** toolbar, click = **Compute**.

RESULTS

Pressure (dl)

1 In the **Model Builder** window, under **Results** click **Pressure (dl)**.

2 In the **Pressure (dl)** toolbar, click **Plot**.

3 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

4 From the **Dataset** list, choose **Study 2: Dual porosity approach/Solution 2 (sol2)**.

5 In the **Pressure (dl)** toolbar, click **Plot**.

The pressure distribution for the homogenized and dual porosity approach differ slightly.

Concentration (tds)

Modify the concentration plot to obtain Figure 4.

Surface 1

1 In the **Model Builder** window, expand the **Concentration (tds)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **Twilight**.

4 From the **Color table transformation** list, choose **Reverse**.

Add a contour plot for the salt distribution from the previous study. This provides a clear picture of the differences between both approaches.

Concentration (tds)

1. In the **Model Builder** window, click **Concentration (tds)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 2: Dual porosity approach/Solution 2 (sol2)**.

Contour 1

1. Right-click **Concentration (tds)** and choose **Contour**.
2. In the **Settings** window for **Contour**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 1: Homogenized approach/Solution 1 (sol1)**.
4. Locate the **Expression** section. In the **Expression** text field, type `c`.
5. Locate the **Levels** section. From the **Entry method** list, choose **Levels**.
6. In the **Levels** text field, type `cs/2`.
7. Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
8. From the **Color** list, choose **Black**.
9. Clear the **Color legend** checkbox.
10. In the **Concentration (tds)** toolbar, click **Plot**.

Concentration (tds)

1. In the **Model Builder** window, click **Concentration (tds)**.
2. In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.
3. From the **Title type** list, choose **Manual**.
4. In the **Title** text area, type `Molar concentration, c (mol/m<sup>3</sup>) and total concentration flux (arrows)`.
5. In the **Concentration (tds)** toolbar, click **Plot**.
6. Click the **Zoom Extents** button in the **Graphics** toolbar.

Precipitation Rate

Lastly, include a plot depicting the precipitation rate. This is an optional element meant to show that the precipitation variability is considered in this model.

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Precipitation Rate` in the **Label** text field.

Global 1

1. Right-click **Precipitation Rate** and choose **Global**.
2. In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Ambient data** > **ampr1.P0_amb - Ambient precipitation rate - m/s**.
3. Locate the **Data** section. From the **Dataset** list, choose **Study 2: Dual porosity approach/ Solution 2 (sol2)**.
4. In the **Precipitation Rate** toolbar, click **Plot**.

STUDY 1: HOMOGENIZED APPROACH

Just in case you want to run the homogenized study again, you can deactivate the dual porosity domain conditions in the solver settings for this study as follows:

Step 1: Time Dependent

1. In the **Model Builder** window, under **Study 1: Homogenized approach** click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.
3. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Domain ODEs and DAEs (dode)**.
4. Select the **Modify model configuration for study step** checkbox.
5. In the tree, select **Component 1 (comp1)** > **Darcy's Law (dl)** > **Dual Porosity Medium 1**.
6. Right-click and choose **Disable**.
7. In the tree, select **Component 1 (comp1)** > **Transport of Diluted Species in Porous Media (tds)** > **Heterogeneous Reactions 1**.
8. Right-click and choose **Disable**.