Model created in COMSOL Multiphysics 6.4

Furrow Irrigation — Dual Permeability

Introduction

This example shows how to set up a model of furrow irrigation in a nonuniform soil column. It employs the Unsaturated Dual Permeability feature, which links two Richards' Equations through a fluid transfer function. This scenario can be regarded as a benchmark problem for dual permeability modeling initially suggested by Ref. 1.

Model Definition

In this example model, water infiltrates from a furrow into a 1.5 m deep soil column over a 6-hour period. The water level in the furrow is kept 10 cm above the soil surface, thus resulting in a constant pressure head boundary condition. At the bottom of the soil profile a zero pressure head boundary condition is applied. Zero-flux flow conditions are imposed on all remaining boundaries.

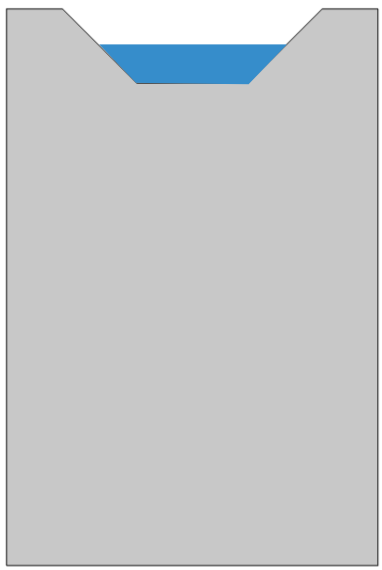

Figure 1: Model setup.

The dual permeability approach is to solve two Richards' equations, one for the macroscopic soil structures referred to as macropores (index i = M) and one for the microscopic soil structures referred to as micropores (index i = m):

$$\theta_i\left(\frac{C_{m,i}}{g}\frac{\partial p_i}{\partial t} + \nabla\left(-\frac{\rho\kappa_i}{\mu}(\nabla p_i - \rho\mathbf{g})\right)\right) = \mp Q_{ip} \tag{1}$$

with θ_i being the respective volume fraction, $C_{m,i}$ the moisture capacity, g the gravitational constant, p_i the pressure in the respective system, κ_i the permeability, ρ and μ the fluid density and dynamic viscosity, respectively, and Q_{ip} represents the interporosity flow where $+Q_{ip}$ indicates flow into the micropores and $-Q_{ip}$ the flow out of the macropores.

Results and Discussion

The pressure distribution within the macropores and micropores, following 6 hours of irrigation, is illustrated in Figure 2. Observations reveal that the pressure in the macropores has nearly stabilized uniformly across the entire depth. However, the pressure distribution within the micropores still exhibits a significant gradient near the furrow, indicating that it has not yet reached a state of equilibrium.

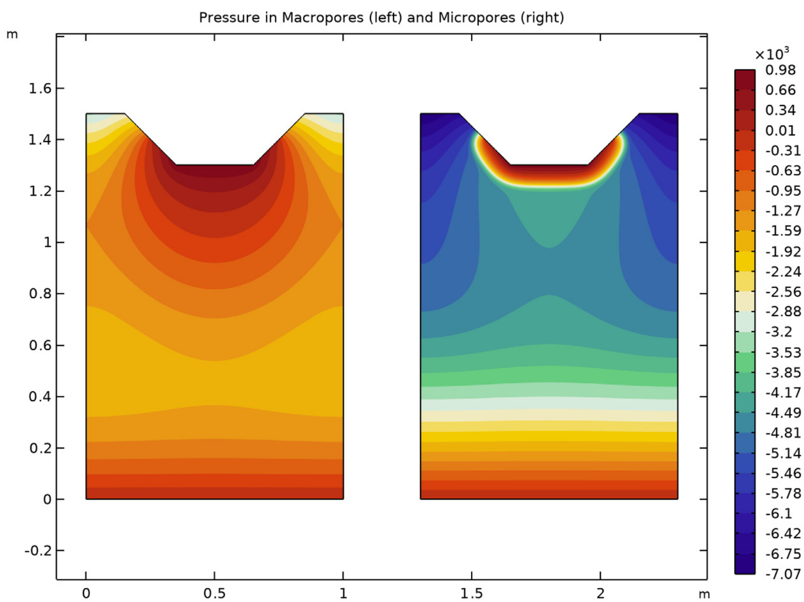

Figure 2: Pressure field in the macropores (left) and micropores (right) after six hours of simulated time.

Similarly, for the effective saturation depicted in Figure 3, the saturation level in the macropores has reached a depth of 30 cm below the furrow, whereas in the micropores it has only penetrated to a depth of 5 cm.

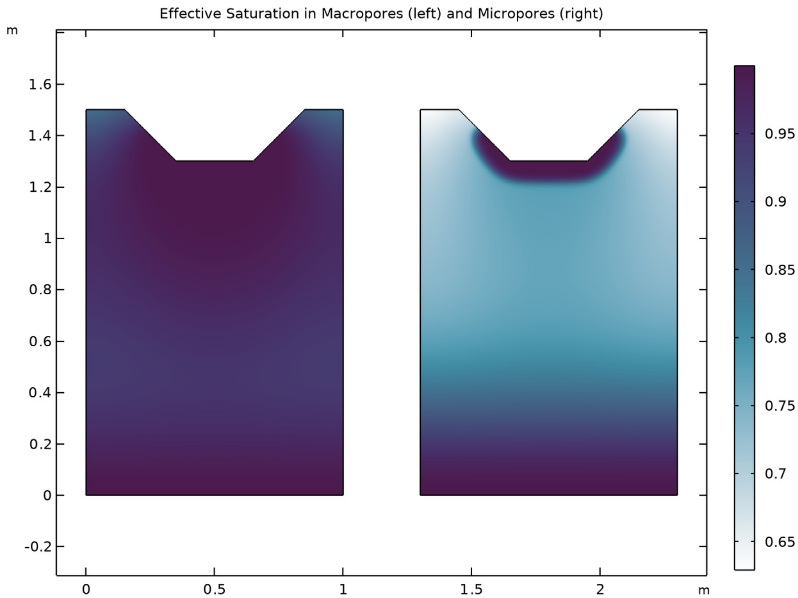

Figure 3: Effective saturation in the macropores (left) and micropores (right) after six hours of simulated time.

Reference

1. T. Vogel and others, “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.

Application Library path: `Subsurface_Flow_Module/Verification_Examples/furrow_irrigation_dual_permeability`

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** > **Richards' Equation (dl)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Time Dependent**.

6 Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Height** text field, type 1.5.

4 Click **Build Selected**.

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)
0.15	1.5
0.25	1.4
0.35	1.3
0.65	1.3
0.75	1.4
0.85	1.5

4 Click **Build Selected**.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **pol1** only.

6 Click **Build Selected**.

Form Union (fin)

In the **Geometry** toolbar, click **Build All**.

GLOBAL DEFINITIONS

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 `furrow_irrigation_dual_permeability_parameters.txt`.

DEFINITIONS

Define an interpolation function for the pressure head in the furrows so that it reflects standing water in the furrows.

Applied pressure head

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

2 Right-click **Definitions** and choose **Functions** > **Interpolation**.

3 In the **Settings** window for **Interpolation**, type `Applied pressure head` in the **Label** text field.

4 Locate the **Definition** section. In the table, enter the following settings:

t	f(t)
1.4	0
1.3	10

5 In the **Function name** text field, type `hp0`.

6 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
hp0	cm

7 In the **Argument** table, enter the following settings:

Argument	Unit
t	m

RICHARDS' EQUATION (DL)

Unsaturated Dual Permeability Medium 1

1 In the **Physics** toolbar, click **Domains** and choose **Unsaturated Dual Permeability Medium**.

2 Select Domain 1 only.

3 In the **Settings** window for **Unsaturated Dual Permeability Medium**, locate the **Interporosity Flow** section.

4 In the α_w text field, type `alpha_w`.

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_f`.

4 From the μ list, choose **User defined**. In the associated text field, type `mu_f`.

Macropores 1

1 In the **Model Builder** window, click **Macropores 1**.

2 In the **Settings** window for **Macropores**, locate the **Volume Fraction** section.

3 In the θ_M text field, type `0.05`.

4 Locate the **Matrix Properties** section. From the $\varepsilon_{p,M}$ list, choose **User defined**. In the associated text field, type `epsilon_p_M`.

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

6 In the $K_{s,M}$ text field, type `Ks_M`.

7 Locate the **Retention Model** section. In the α text field, type `alpha`.

8 In the n text field, type `n`.

9 In the l text field, type `l`.

10 In the θ_r text field, type `theta_r`.

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 $\varepsilon_{p,m}$ list, choose **User defined**. In the associated text field, type `epsilon_p_m`.

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

5 In the $K_{s,m}$ text field, type `Ks_m`.

The retention model is the same as in the Macropores.

Initial Values 1

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

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

3 Click the **Hydraulic head** button.

Pressure Head 1

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

2 Select Boundaries 5–7 only.

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

4 In the H_{p0} text field, type `hp0(y)`.

Pressure Head 2

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

2 Select Boundary 2 only.

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extremely fine**.

Free Triangular 1

In the **Model Builder** window, right-click **Free Triangular 1** and choose **Build Selected**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** 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 **h**.

4 In the **Output times** text field, type `range(0,0.5,6)`.

5 In the **Study** toolbar, click **Compute**.

RESULTS

Pressure Head

1 In the **Model Builder** window, expand the **Results** > **Pressure (dl)** node, then click **Pressure (dl)**.

2 In the **Settings** window for **2D Plot Group**, type `Pressure Head` in the **Label** text field.

Surface

1 In the **Model Builder** window, click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Richards' Equation** > **Velocity and pressure** > **dl.Hp - Pressure head - m**.

3 In the **Pressure Head** toolbar, click **Plot**.

Include pressure and saturation plots for the macro- and micropores using the available result templates.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Richards' Equation** > **Pressure in Macro- and Micropores (dl)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the tree, select **Study 1/Solution 1 (sol1)** > **Richards' Equation** > **Effective Saturation in Macro- and Micropores (dl)**.

6 Click the **Add Result Template** button in the window toolbar.

7 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

Compare the two images with Figure 2 and Figure 3.