



Wicking in a Paper Strip

Introduction

Wicking is the phenomenon that occurs when a dry porous material is put into contact with a fluid: it will absorb fluid due to capillary forces. The absorption will continue until an equilibrium is reached where the gravitational forces balance capillary forces. This example, inspired by [Ref. 1](#), models wicking in a paper-like porous medium.

Model Definition

The model is set up as a 2D model of a paper strip with a rectangular geometry of 12 cm length and 1.5 cm width. The thickness of the strip is defined as 1 mm.

The model solves for air saturation and water pressure. The different material parameters used in this model are taken from [Ref. 1](#) and are presented in [Table 1](#).

TABLE 1: MATERIAL PROPERTIES FOR WATER, AIR, AND THE POROUS MATERIAL.

Material property	Value	Unit
Density of water	1000	kg/m ³
Density of air	1	kg/m ³
Viscosity of water	0.001	Pa·s
Viscosity of air	$1.76 \cdot 10^{-5}$	Pa·s
Porosity	0.6	l
Permeability	$5.808 \cdot 10^{-14}$	m ²
Pore radius	$8.8 \cdot 10^{-7}$	m
Surface tension of water	0.0723	N/m
Contact angle	0	l
Entry capillary pressure	$1.643 \cdot 10^5$	Pa
Pore size distribution index	2	l

The paper strip is initially filled with air with an initial water saturation of 0.01. The Brooks and Corey model for capillary pressure and relative permeabilities is used for the time-dependent simulation. At the side boundaries of the paper strip no water flow and is allowed. The boundary conditions for bottom and top boundaries for Darcy's law are the atmospheric pressure for water phase at the bottom and the hydrostatic atmospheric pressure minus the capillary pressure at the top. For the phase transport, there is no flux assumed for the air phase at the bottom boundary. At the top boundary, a mass flux for the air phase is defined which results from the pressure gradient given in the Darcy model. In the section [Notes About the COMSOL Implementation](#) it is specified how an accurate mass flux can be calculated in COMSOL Multiphysics.

MODEL EQUATIONS

This example uses the multiphysics coupling Multiphase Flow in Porous Media, which couples the Darcy's Law and Phase Transport in Porous Media interfaces.

The Phase Transport in Porous Media interface follows separate equations for the volume fraction s_i of the wetting or nonwetting fluid i :

$$\frac{\partial}{\partial t}(\varepsilon_p \rho_i s_i) + \nabla \cdot \left(-\rho_i \kappa \frac{\kappa_{ri}}{\mu_i} (\nabla p_i - \rho_i \mathbf{g}) \right) = Q_i = 0 \quad (1)$$

Here, ε_p is the porosity, κ is the permeability (m^2), κ_{ri} is the relative permeability (a function of saturation for a given fluid), μ_i is the fluid's dynamic viscosity (Pa·s), p_i is the pressure (Pa), and ρ_i is the fluid density (kg/m^3) of phase i .

As the sum of the volume fractions of the two phases is 1, the remaining volume fraction is computed from

$$s_1 = 1 - s_2 \quad (2)$$

The capillary pressure p_c is calculated as a function of the saturation of the wetting phase s_w (which is s_2 in the model) and the entry capillary pressure p_{ec} . By using the Brooks and Corey model, the capillary pressure is given by:

$$p_c = p_{ec} \frac{1}{(s_w)^{1/\lambda_p}} \quad (3)$$

where λ_p is the pore distribution index.

The relative permeabilities for the wetting and nonwetting phases, based on the Brooks and Corey model, are given by

$$\kappa_{rs_w} = (\overline{s_w})^{(3 + 2/\lambda_p)} \quad (4)$$

$$\kappa_{rs_n} = \overline{s_n}^{-2} (1 - (1 - \overline{s_n})^{(1 + 2/\lambda_p)}) \quad (5)$$

The Darcy's Law interface combines Darcy's law with the continuity equation:

$$\frac{\partial}{\partial t}(\rho \varepsilon_p) + \nabla \cdot \rho \left[-\frac{\kappa}{\mu} (\nabla p) \right] = 0 \quad (6)$$

where density ρ and dynamic viscosity μ are average values of the two wetting and nonwetting fluids.

Results and Discussion

A transient simulation of 10 minutes is performed. After 600 s, the paper strip is already soaked with water. [Figure 1](#) shows the water saturation for different times.

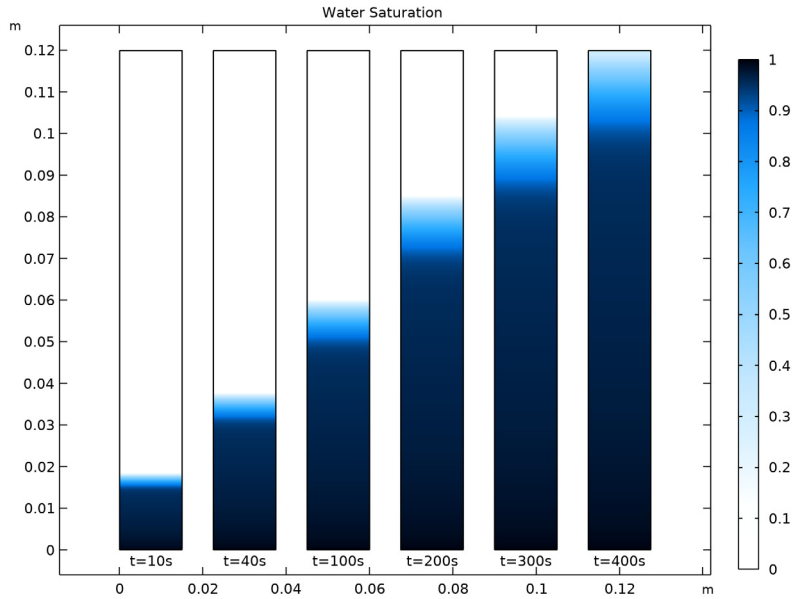


Figure 1: Water Saturation in the paper strip after 10, 40, 100, 200, 300, and 400 s.

The water saturation in the paper strip can also be displayed for a full 3D geometry as shown in [Figure 2](#).

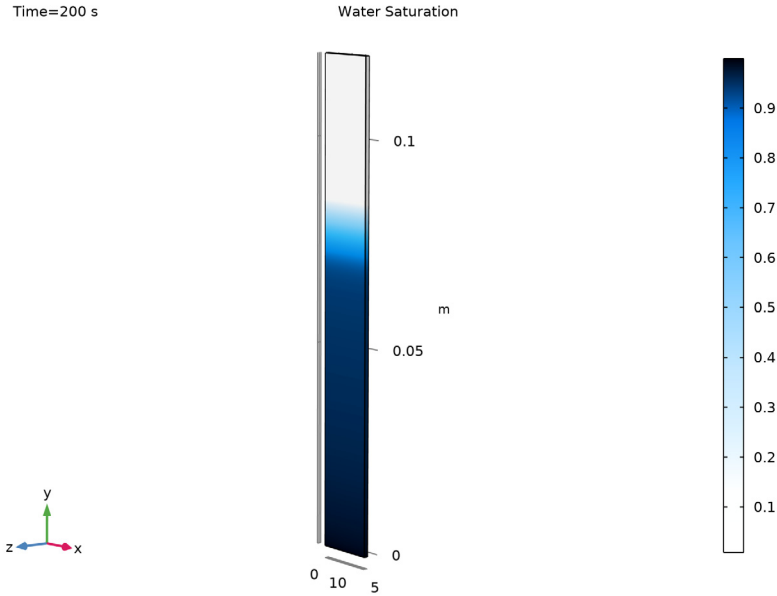


Figure 2: Water Saturation (3D) after 200 s.

Figure 3 shows the amount of water that is absorbed by the paper strip as a function of time. It is calculated from the height of the liquid front H_{lf} . In this example model, the simulation values (solid line) are compared with those calculated from the analytical expression given by the Lucas-Washburn equation (dashed line), where H_{lf} is defined as

$$H_{lf} = \sqrt{\frac{\gamma R_c \cos \Theta}{2\mu_w}} \quad (7)$$

with γ the surface tension, R_c the pore radius, Θ the contact angle and μ_w the dynamic viscosity of water. Figure 3 shows that the two curves are in good agreement. Note that both Brooks and Corey model and Lucas-Washburn equation are analytical expression that matches experimental results and analytical expressions. You can directly enter the capillary pressure curve if you have experimental data for it.

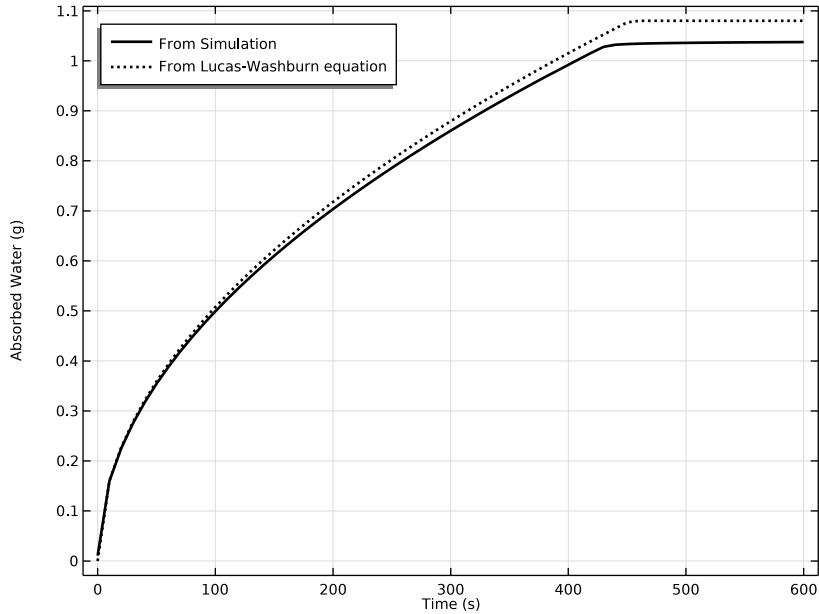


Figure 3: Water uptake versus time. Solid line: numerical solution. Dashed line: analytical expression given by the Lucas-Washburn equation.

Notes About the COMSOL Implementation

In this model, the top boundary condition for the Phase Transport in Porous Media interface is a mass flux condition where the mass flux results from the pressure gradient computed using Darcy's Law. To calculate an accurate mass flux, weak constraints are used.

When using weak constraints in an interface, the Lagrange multipliers are additional dependent variables in that interface. The sign of the Lagrange multiplier is the same as the one used when applying the corresponding quantity explicitly in a flux condition.

For postprocessing, it is important to know that in (axially) symmetric models the Lagrange multipliers are not equal to the reaction flux per area but rather per length and full revolution.

For more information about weak constraints, the calculation of accurate fluxes, and Lagrange multipliers, see the sections *Boundary Conditions* and *Computing Accurate Fluxes* in the *COMSOL Multiphysics Reference Manual*.

Reference


1. R. Masoodi and K.M. Pillai, “Darcy’s Law-Based Model for Wicking in Paper-Like Swelling Porous Media,” *AIChE J.*, vol. 56, pp 2257–2267, 2010.

Application Library path: Porous_Media_Flow_Module/Fluid_Flow/wicking_paper




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>Multiphase Flow in Porous Media**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies>Time Dependent**.
- 6 Click  **Done**.

GLOBAL DEFINITIONS

Parameters 1

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

Define the parameters for the geometry and the material properties as follows:


- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
L0	12[cm]	0.12 m	Paper strip height
W0	L0/8	0.015 m	Paper strip width

Name	Expression	Value	Description
th	1[mm]	0.001 m	Paper strip thickness
gamma	0.0723[N/m]	0.0723 N/m	Surface tension
theta	0	0	Contact angle
Rc	8.8e-7[m]	8.8E-7 m	Pore radius
pec	$2 * \text{gamma} * \cos(\text{theta}) / \text{Rc}$	1.6432E5 N/m ²	Entry capillary pressure
lp	2	2	Pore size distribution index
por	0.6	0.6	Porosity
K	$\text{por} / 8 * \text{Rc}^2$	5.808E-14 m ²	Permeability
rho_air	1[kg/m ³]	1 kg/m ³	Air density
rho_water	1e3[kg/m ³]	1000 kg/m ³	Water density
mu_air	1.76e-5[Pa*s]	1.76E-5 Pa*s	Air viscosity
mu_water	0.001[Pa*s]	0.001 Pa*s	Water viscosity


DEFINITIONS

Step 1 (step1)

- 1 In the **Home** toolbar, click  **Functions** and choose **Local>Step**.
- 2 In the **Settings** window for **Step**, locate the **Parameters** section.
- 3 In the **Location** text field, type 0.9.
- 4 In the **From** text field, type 1.
- 5 In the **To** text field, type 0.
- 6 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type 0.2.
This step function will be used to define the upper boundary condition for Darcy's Law. It will ensure that the capillary pressure at the top is set to zero as soon as the paper is soaked with water.

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 **Width** text field, type W0.
- 4 In the **Height** text field, type L0.

5 Click  **Build All Objects**.

MATERIALS

Define the materials in the next step. Introduce them as empty material nodes, first. As soon as the physics has been defined, the material node menu will show you which properties are needed for the simulation. You can then just fill in the values defined in the **Parameters** list.

Porous Material 1 (pmat1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Porous Material**.

Water

- 1 Right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, type Water in the **Label** text field.

Air

- 1 Right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, type Air in the **Label** text field.


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 **Physical Model** section.
- 3 In the d_z text field, type th.
- 4 Click to expand the **Discretization** section. From the **Pressure** list, choose **Linear**.

Pressure 1


- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Pressure**.
- 2 Select Boundary 2 only.

Pressure 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Pressure**.
- 2 Select Boundary 3 only.
- 3 In the **Settings** window for **Pressure**, locate the **Pressure** section.
- 4 In the p_0 text field, type $-(\text{phtr.pc_s2}) * \text{step1}(s1) - \text{rho_air} * L0 * g_const$ to calculate the hydrostatic pressure minus the capillary pressure at the upper boundary.

The capillary pressure is set to zero when the paper strip is soaked with water by multiplying it with the smoothed step function defined earlier in the model.

The flux at this boundary will be needed as a boundary condition for the **Phase Transport in Porous Media** interface. Therefore, weak constraints are used to assure an accurate flux calculation (see the section [Notes About the COMSOL Implementation](#)). To activate them, follow the instructions below:

- 5 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.
- 7 Click **OK**.
- 8 In the **Settings** window for **Pressure**, click to expand the **Constraint Settings** section.
- 9 From the **Apply reaction terms** on list, choose **Individual dependent variables**.
- 10 Select the **Use weak constraints** check box.

PHASE TRANSPORT IN POROUS MEDIA (PHTR)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Phase Transport in Porous Media (phtr)**.
- 2 In the **Settings** window for **Phase Transport in Porous Media**, locate the **Gravity Effects** section.
- 3 Select the **Include gravity** check box.

Phase and Porous Media Transport Properties 1


- 1 In the **Model Builder** window, under **Component 1 (comp1)> Phase Transport in Porous Media (phtr)** click **Phase and Porous Media Transport Properties 1**.
- 2 In the **Settings** window for **Phase and Porous Media Transport Properties**, locate the **Capillary Pressure** section.
- 3 From the **Capillary pressure model** list, choose **Brooks and Corey**.
- 4 Click to expand the **Equation** section. Locate the **Capillary Pressure** section. In the p_{ec} text field, type p_{ec} .
- 5 In the λ_p text field, type $1p$.
- 6 Locate the **Phase 1 Properties** section. From the **Fluid s1** list, choose **Water (mat1)**.
- 7 Locate the **Phase 2 Properties** section. From the **Fluid s2** list, choose **Air (mat2)**.

Initial Values 1

- 1 In the **Model Builder** window, click **Initial Values 1**.

- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 In the $s_{0,s2}$ text field, type 0.99.

Mass Flux 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Mass Flux**.
- 2 In the **Settings** window for **Mass Flux**, locate the **Mass Flux** section.
- 3 Select the **Phase s2** check box.
- 4 Select Boundary 3 only.
- 5 In the $q_{0,s2}$ text field, type $p_{1m}[\text{kg/m/s}]/\text{th}$. Here p_{1m} is the Lagrange multiplier representing the flux at the boundary (see the section [Notes About the COMSOL Implementation](#)). It was introduced as an additional variable to solve for when the weak constraints were activated for this boundary in the **Darcy's Law** interface. As the Lagrange multiplier does not take into account the thickness of the paper strip the flux has to be divided by it.

Now, you can fill in the missing material properties

MATERIALS

Porous Material 1 (pmat1)

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Porous Material 1 (pmat1)**.
- 2 In the **Settings** window for **Porous Material**, locate the **Porosity** section.
- 3 In the ϵ_p text field, type por.
- 4 Locate the **Homogenized Properties** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Permeability	κ_{iso} ; $\kappa_{\text{p}ii} = \kappa_{\text{iso}}$, $\kappa_{\text{p}ij} = 0$	K	m ²	Basic
Porosity	epsilon	1	1	Porous model

Water (mat1)

- 1 In the **Model Builder** window, click **Water (mat1)**.
- 2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rho_water	kg/m ³	Basic
Dynamic viscosity	mu	mu_water	Pa·s	Basic

Air (mat2)

1 In the **Model Builder** window, click **Air (mat2)**.


2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rho_air	kg/m ³	Basic
Dynamic viscosity	mu	mu_air	Pa·s	Basic

MESH 1

Mapped 1

In the **Mesh** toolbar, click  **Mapped**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 2 only.

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

4 In the **Number of elements** text field, type 1.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.


2 Select Boundaries 1 and 4 only.

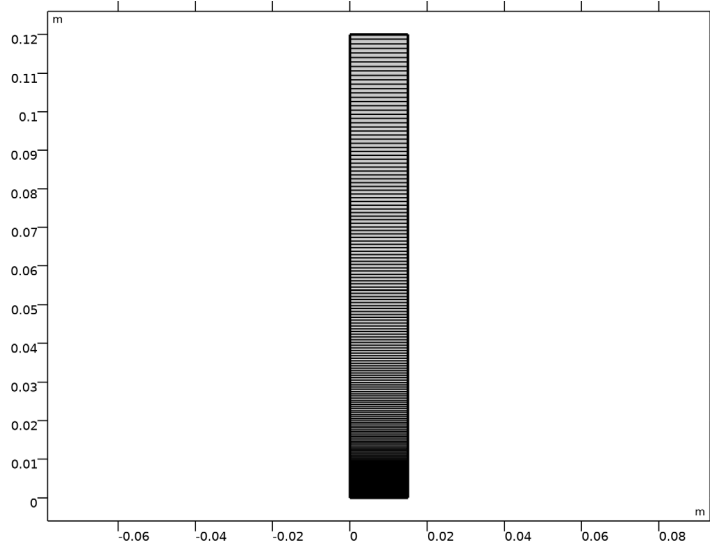
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 200.

6 In the **Element ratio** text field, type 1000.

- 7 Click  **Build All**. The dense mesh at the bottom is needed to resolve the very steep saturation gradient in the initial phase of the process.




STUDY 1


Now set up the study. The paper strip is fully soaked with water after 10 minutes so this is the end time to choose for the simulation. As there are very steep gradients in the volume fractions of water and air, make sure that the initial time step is small enough by following the instructions below.

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 In the **Output times** text field, type range (0, 10, 600).

Solution 1 (sol1)


- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.
- 3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 4 Select the **Initial step** check box.
- 5 In the associated text field, type 0.0001.

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


RESULTS

A surface plot of the volume fraction of water (s1) and the pressure is generated by default. To group the different surface plots of the water saturation directly in one postprocessing window as shown in [Figure 1](#) follow the steps below:

Water Saturation (grouped)

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **2D Plot Group**.
- 2 In the **Settings** window for **2D Plot Group**, type Water Saturation (grouped) in the **Label** text field.
- 3 Locate the **Data** section. From the **Time (s)** list, choose **10**.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 5 In the **Title** text area, type Water Saturation.
- 6 Clear the **Parameter indicator** text field.

Surface 1

- 1 Right-click **Water Saturation (grouped)** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
- 4 From the **Time (s)** list, choose **10**.
- 5 Locate the **Coloring and Style** section. From the **Color table** list, choose **JupiterAuroraBorealis**.
- 6 From the **Color table transformation** list, choose **Reverse**.
- 7 Click to expand the **Range** section. Select the **Manual color range** check box.
- 8 In the **Maximum** text field, type 1.
- 9 In the **Water Saturation (grouped)** toolbar, click  **Plot**.
Use the **Plot Array** functionality to create multiple plots next to each other.

Water Saturation (grouped)

- 1 In the **Model Builder** window, click **Water Saturation (grouped)**.
- 2 In the **Settings** window for **2D Plot Group**, click to expand the **Plot Array** section.
- 3 Select the **Enable** check box.
- 4 In the **Relative padding** text field, type 0.5.

Surface 2

- 1 In the **Model Builder** window, under **Results>Water Saturation (grouped)** right-click **Surface 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **40**.
- 4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Surface 3

- 1 Right-click **Surface 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **100**.

Surface 4

- 1 Right-click **Surface 3** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **200**.

Surface 5

- 1 Right-click **Surface 4** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **300**.


Surface 6

- 1 Right-click **Surface 5** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **400**.

Water Saturation (grouped)

In the **Model Builder** window, click **Water Saturation (grouped)**.

Table Annotation 1

- 1 In the **Water Saturation (grouped)** toolbar, click  **More Plots** and choose **Table Annotation**.
- 2 In the **Settings** window for **Table Annotation**, locate the **Data** section.
- 3 From the **Source** list, choose **Local table**.


4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
$W_0/2$	0	t=10s
$2*W_0$	0	t=40s
$3*W_0+W_0/2$	0	t=100s
$5*W_0$	0	t=200s
$6.5*W_0$	0	t=300s
$8*W_0$	0	t=400s

5 Locate the **Coloring and Style** section. Clear the **Show point** check box.

6 From the **Anchor point** list, choose **Upper middle**.

7 In the **Water Saturation (grouped)** toolbar, click  **Plot**.

8 Click the  **Zoom Extents** button in the **Graphics** toolbar.

To reproduce [Figure 2](#) create a 3D dataset and use this to plot the water saturation by following the instructions below.

Extrusion 2D 1


1 In the **Results** toolbar, click  **More Datasets** and choose **Extrusion 2D**.

2 In the **Settings** window for **Extrusion 2D**, locate the **Extrusion** section.

3 In the **z maximum** text field, type `th`.

4 Click  **Plot**.

3D Plot Group 4

1 In the **Results** toolbar, click  **3D Plot Group**.

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

3 From the **Time (s)** list, choose **200**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Water Saturation`.

6 In the **Parameter indicator** text field, type `Time=eval(t) s`.

Surface 1


1 Right-click **3D Plot Group 4** and choose **Surface**.

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

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

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

Water Saturation 3D

- 1 In the **Model Builder** window, under **Results** click **3D Plot Group 4**.
- 2 In the **Settings** window for **3D Plot Group**, type Water Saturation 3D in the **Label** text field.
- 3 In the **Water Saturation 3D** toolbar, click  **Plot**.

GLOBAL DEFINITIONS

To compare the simulation results with theoretical values based on the Lucas-Washburn equation (see [Figure 3](#)), an additional variable, the height of the liquid front H_{1f} , is introduced (as in [Equation 7](#)) and used to compute the absorbed amount of water as a function of time.

Variables 1

- 1 In the **Model Builder** window, right-click **Global 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
H_1f	$\sqrt{(\gamma R_c \cos(\theta) t) / (2 \mu_{\text{water}})}$	m	Height of liquid front


DEFINITIONS

Integration 1 (intop1)

- 1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Integration**.
- 2 Select Domain 1 only.

STUDY 1

Now you have to update your solution to get the new variable for all time steps.

- 1 In the **Study** toolbar, click  **Update Solution**.

RESULTS

Global Evaluation 1

- 1 In the **Results** toolbar, click  **Global Evaluation**.
- 2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
$\text{intop1}(s1)*th*\rho_{\text{water}}*por$	g	Absorbed water, from simulation
$\text{min}(L0,H_{1f})*W0*th*\rho_{\text{water}}*por$	g	Absorbed water, from Lucas-Washburn equation

4 Click  **Evaluate**.

TABLE

- 1 Go to the **Table** window.
- 2 Click **Table Graph** in the window toolbar.

RESULTS

Water Uptake

- 1 In the **Model Builder** window, under **Results** click **ID Plot Group 5**.
- 2 In the **Settings** window for **ID Plot Group**, type *Water Uptake* in the **Label** text field.


Table Graph 1

- 1 In the **Model Builder** window, click **Table Graph 1**.
- 2 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.
- 3 Find the **Line style** subsection. From the **Line** list, choose **Cycle**.
- 4 From the **Color** list, choose **Black**.
- 5 In the **Width** text field, type 2.
- 6 Click to expand the **Legends** section. Select the **Show legends** check box.
- 7 From the **Legends** list, choose **Manual**.
- 8 In the table, enter the following settings:

Legends
From Simulation
From Lucas-Washburn equation

Water Uptake

- 1 In the **Model Builder** window, click **Water Uptake**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Legend** section.
- 3 From the **Position** list, choose **Upper left**.

- 4 In the **Water Uptake** toolbar, click  **Plot**.
- 5 Locate the **Plot Settings** section. Select the **y-axis label** check box.
- 6 In the associated text field, type Absorbed Water (g).
Compare this plot to [Figure 3](#).

