Created in COMSOL Multiphysics 6.2

Moisture Transport in a Paperboard Roll

Introduction

Paperboard is made of wood fibers, which can absorb water. Water absorption changes the mechanical properties of the paperboard, and it might affect the quality of the final product due to moisture-induced deformations. This deformation is caused by the pressure acting on the pore walls. Paperboard is typically modeled as an orthotropic material since the cellulose fibers are mostly aligned along the grain direction, or so-called machine direction. The arrangement of the fibers affects also the transport properties.

This example shows how to model moisture transport in a paperboard roll when the relative humidity in the room changes during 90 days storage.

Model Definition

The model geometry consists of a hollow paperboard cylinder 70 cm long and 40 cm thick, lying on a 20 cm diameter shaft. A 2D axisymmetric geometry is used, and only half of it is modeled due to symmetry. A roller boundary condition is used on the side lying on the shaft. The temperature is held fixed at 293.15 K and the ambient relative humidity is increased from 5% to 85% in 1 day and kept constant for 90 days. The paperboard absorbs the moisture in the surrounding air, causing hygroscopic swelling. The pore pressure p_f in the paperboard is computed from:

$$p_\mathrm{f} = s_\mathrm{l} p_\mathrm{l} + (1 - s_\mathrm{l}) p_\mathrm{mA}$$

where s_l is the liquid saturation, p_l is the liquid pressure, $s_\mathrm{m} = 1 - s_\mathrm{l}$ is the moist air saturation, and p_mA is the moist air absolute pressure. The relation between the relative humidity, ϕ_w, and the water content, w_c, is described by the sorption isotherm

$$w_\mathrm{c} = -\frac{1}{B}\log\left(-\frac{RT\log\phi_\mathrm{w}}{A}\right)$$

here, A and B are fitting parameters (6.232E3 J/mol and 20.56 respectively). The moist air permeability, κ_m, is described using the following model

$$\kappa_\mathrm{m} = \kappa\left(\frac{\varepsilon_\mathrm{p} s_\mathrm{m}}{\varepsilon_\mathrm{p0} s_\mathrm{m0}}\right)^a \cdot \frac{1 - \varepsilon_\mathrm{p0} s_\mathrm{m0}}{1 - \varepsilon_\mathrm{p} s_\mathrm{m}}$$

where s_m is the moist air saturation, ε_p is the porosity, κ is the reference permeability, and the exponent a are obtained from experiments. Variables with a 0 subscript refer to their initial value.

Different properties and material parameters are used in the local directions, namely the machine direction (MD), cross-machine direction (CD), and the through-thickness direction (ZD).

The parameters are listed in the tables below:

TABLE 1: MODEL PARAMETERS

Parameter	Value
Porosity	0.5
Density of paperboard	15000 kg/m^3
Ambient pressure	1 atm
Ambient temperature	293.15 K

TABLE 2: TRANSPORT PARAMETERS (Ref. 1)

PARAMETER	VALUE
Liquid relative permeability	0
Moist air relative permeability	1
Permeability, k (ZD)	4.3646E-15 m^2
Exponent parameter, a (ZD)	4.18
Permeability, k (CD)	1.9405E-13 m^2
Exponent parameter, a (CD)	2.2
Vapor diffusion (ZD)	1.2929E-12 s
Vapor diffusion (CD)	7.1295E-11 s

TABLE 3: ELASTIC MATERIAL PROPERTIES (Ref. 2)

PARAMETER	VALUE
Young's modulus (MD)	3.2 GPa
Young's modulus (CD)	2 GPa
Young's modulus (ZD)	0.016 GPa
Shear modulus (MD-CD)	1 GPa
Shear modulus (CD-ZD)	0.057 GPa
Shear modulus (MD-ZD)	0.058 GPa
Poisson's ratio (MD-CD)	0.34
Poisson's ratio (CD-ZD)	0.01
Poisson's ratio (MD-ZD)	0.01

Results and Discussion

Figure 1 and Figure 2 show the relative humidity after 90 days. Due to the permeability and diffusivity anisotropy, the moisture penetrates faster in the machine direction (vertical diction) than in the through-thickness direction (radial direction).

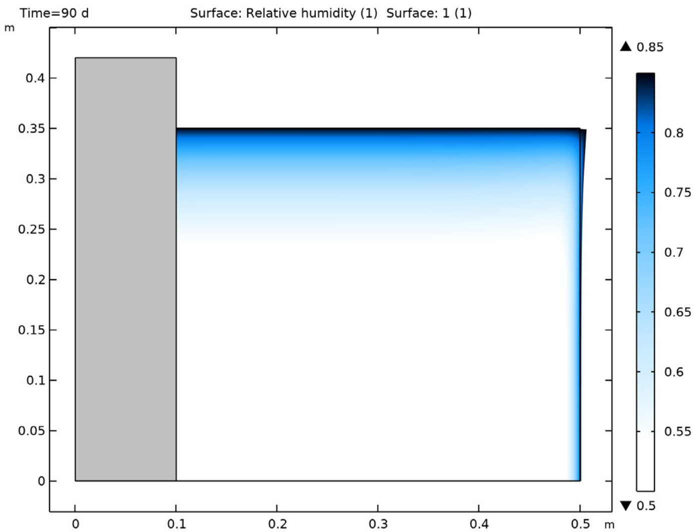

Figure 1: Relative humidity in the deformed configuration after 90 days.

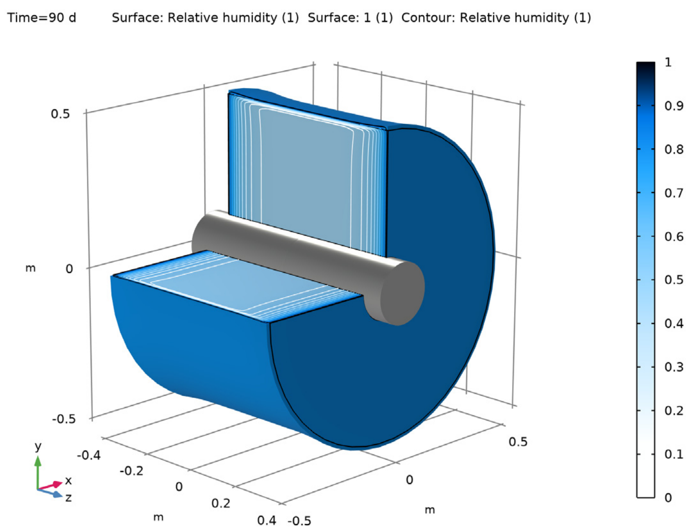

Figure 2: Relative humidity in the deformed configuration after 90 days.

Figure 3 and Figure 4 shows the moisture content through the paperboard in the machine and through-thickness directions at various times.

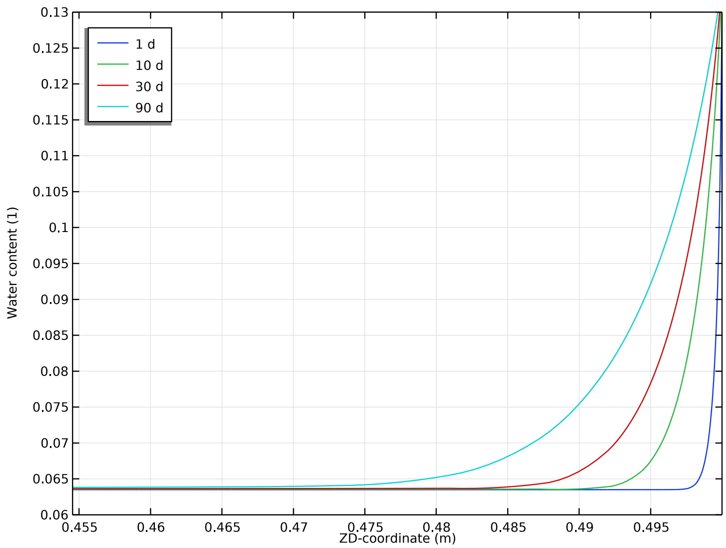

Figure 3: Water content distribution in the paperboard through-thickness direction (radial direction) at various instants.

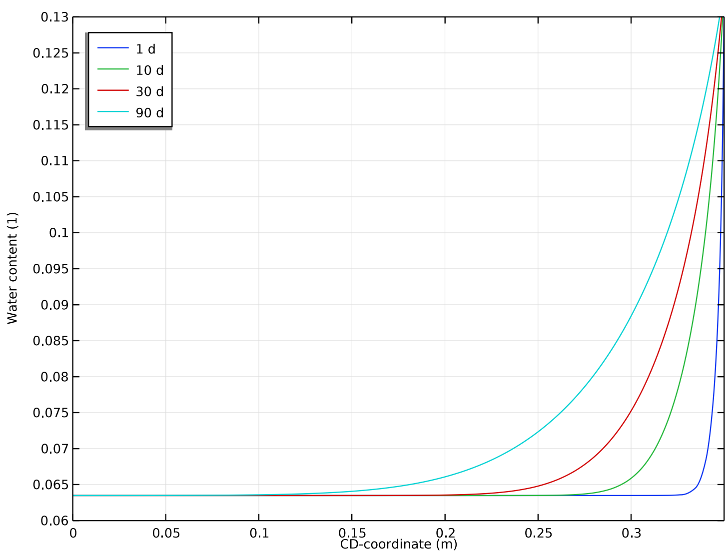

Figure 4: Water content distribution in the paperboard machine direction (vertical, or axial direction) at various instants.

The roll expands mainly in the radial direction because of the lower stiffness of the paperboard in the radial direction. Figure 5 shows that the maximum swelling deformation reaches about 6 mm after 90 days. The corner expands more than the bulk material because of the moisture influx contribution from the top and lateral sides.

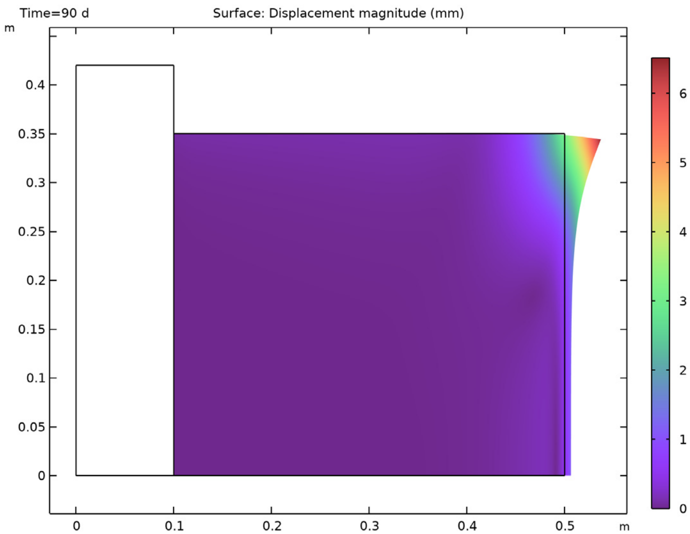

Figure 5: Displacement magnitude and deformation (amplified) after 90 days.

Notes About the COMSOL Implementation

The ambient relative humidity, pressure, and temperature are defined in the **Ambient Properties** node which can be linked to the boundary conditions in the Moisture Transport in Solids interface.

The sorption isotherm and the permeability can be defined using analytic functions under the **Porous Material** node.

The **Unsaturated Poroelasticity** multiphysics coupling is used to model moisture-induced swelling.

References

1. M. Alexandersson, H. Askfelt, and M. Ristinmaa, “Triphasic Model of Heat and Transport with Internal Mass Exchange in Paperboard,” *Transp. Porous Med.*, vol. 112, pp. 381–408, 2016. https://doi.org/10.1007/s11242-016-0651-9.

2. J. Ran and C. Liu, "Modeling of the Stiffness of Corrugated Cardboard Considering Material Nonlinear Effect," *J. Phys.:Conf. Ser.* 1187, 032069. https://doi.org/10.1088/1742-6596/1187/3/032069.

Application Library path: `Structural_Mechanics_Module/Hygroscopic_Swelling/paperboard_roll`

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 Axisymmetric**.

2 In the **Select Physics** tree, select **Structural Mechanics>Poroelasticity>Unsaturated Poroelasticity**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Model Parameters

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 `paperboard_roll_parameters.txt`.

5 In the **Label** text field, type `Model Parameters`.

COMPONENT 1 (COMP1)

You can rename the frame coordinates using the paperboard convention: machine direction (md), cross-machine direction (cd), and through-thickness direction (zd). This can be useful when inserting user-defined material properties.

1. In the **Model Builder** window, click **Component 1 (comp1)**.
2. In the **Settings** window for **Component**, locate the **Frames** section.
3. Find the **Spatial frame coordinates** subsection. In the table, enter the following settings:

First	Second	Third
`zd`	`md`	`cd`

4. Find the **Material frame coordinates** subsection. In the table, enter the following settings:

First	Second	Third
`ZD`	`MD`	`CD`

GEOMETRY 1

Paper

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 `Ro-Ri`.
4. In the **Height** text field, type `H/2`.
5. Locate the **Position** section. In the **zd** text field, type `Ri`.
6. In the **Label** text field, type `Paper`.

Holder

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 `Ri`.
4. In the **Height** text field, type `1.2*H/2`.
5. In the **Label** text field, type `Holder`.

Form Union (fin)

1. In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.
2. In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

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

DEFINITIONS

Boundaries Facing Ambient

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 7 and 8 only.

5 In the **Label** text field, type `Boundaries Facing Ambient`.

Ambient Relative Humidity

1 In the **Definitions** toolbar, click **Piecewise**.

2 In the **Settings** window for **Piecewise**, locate the **Definition** section.

3 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	t_r	phi_init +(phi_end-phi_init)*x/t_r
t_r	t_end	phi_end

4 Locate the **Units** section. In the **Arguments** text field, type `s`.

5 In the **Function** text field, type `1`.

6 In the **Function name** text field, type `phiw_fun`.

7 In the **Label** text field, type `Ambient Relative Humidity`.

Add **Ambient Properties** node to define the ambient temperature, pressure and relative humidity. Then the boundary conditions can link directly to this node.

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 Conditions** section.

3 In the T_{amb} text field, type `T_amb`.

4 In the ϕ_{amb} text field, type `phiw_fun(t)`.

MATERIALS

Porous Material 1 (pmat1)

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

Water Content

1 In the **Model Builder** window, expand the **Porous Material 1 (pmat1)** node.

2 Right-click **Component 1 (comp1)>Materials>Porous Material 1 (pmat1)>Basic (def)** and choose **Functions>Analytic**.

3 In the **Settings** window for **Analytic**, locate the **Definition** section.

4 In the **Expression** text field, type `-(1/B)*log(-R_const*T_amb/A*log(phi))`.

5 In the **Arguments** text field, type `phi`.

6 Locate the **Units** section. In the **Function** text field, type `1`.

7 In the table, enter the following settings:

Argument	Unit
phi	1

8 In the **Function name** text field, type `wc_int`.

9 In the **Label** text field, type `Water Content`.

Moist Air Permeability

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, locate the **Definition** section.

3 In the **Expression** text field, type `k*(epsilonp*sm/(por*0.9))^a*(1-por*0.9)/(1-epsilonp*sm)`.

4 In the **Arguments** text field, type `k, a, sm, epsilonp`.

5 Locate the **Units** section. In the **Function** text field, type `m^2`.

6 In the table, enter the following settings:

Argument	Unit
k	m^2
a	1
sm	1
epsilonp	1

7 In the **Function name** text field, type `kappa_fun`.

8 In the **Label** text field, type `Moist Air Permeability`.

Porous Material 1 (pmat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Porous Material 1 (pmat1)**.

2 Select Domain 2 only.

3 In the **Settings** window for **Porous Material**, locate the **Homogenized Properties** section.

4 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Water content	w_c	`wc_int(phi)*rho_s*(1-por)`	kg/m³	Basic
Permeability	{kappa11, kappa22, kappa33} ; kappaij = 0	`{kappa_fun(Km_zd, a_zd, mts.sm, mts.porosity), 0, kappa_fun(Km_cd, a_cd, mts.sm, mts.porosity)}`	m²	Basic

Solid 1 (pmat1.solid1)

Right-click **Porous Material 1 (pmat1)** and choose **Solid**.

MOISTURE TRANSPORT IN SOLIDS (MTS)

Select Domain 2 only.

MATERIALS

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials> Porous Material 1 (pmat1)** click **Solid 1 (pmat1.solid1)**.

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

3 In the θ_s text field, type `1-por`.

Porous Material 1 (pmat1)

1 In the **Model Builder** window, click **Basic (def)**.

2 In the **Settings** window for **Basic**, locate the **Model Inputs** section.

3 Click **Select Quantity**.

4 In the **Physical Quantity** dialog box, type `relative` in the text field.

5 Click **Filter**.

6 In the tree, select **General>Relative humidity (1)**.

7 Click **OK**.

MOISTURE TRANSPORT IN SOLIDS (MTS)

Liquid Water 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Moisture Transport in Solids (mts)>Porous Medium 1** click **Liquid Water 1**.

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

3 In the text field, type `krl`.

Moist Air 1

1 In the **Model Builder** window, click **Moist Air 1**.

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

3 In the text field, type `krm`.

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

5 In the D_v table, enter the following settings:

tau_zd*(R_const*T_amb/mts.M_v)	0
0	tau_cd*(R_const*T_amb/mts.M_v)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Moisture Transport in Solids (mts)** click **Initial Values 1**.

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

3 In the ϕ_w text field, type `phi_init`.

Moisture Content and Pressure 1

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

You can link the boundaries condition directly to the **Ambient Properties** node defined before.

2 In the **Settings** window for **Moisture Content and Pressure**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Boundaries Facing Ambient**.

4 Locate the **Moisture Content and Pressure** section. From the ϕ_{w0} list, choose **Ambient relative humidity (ampr1)**.

5 From the p_{m0} list, choose **Ambient absolute pressure (ampr1)**.

Symmetry 1

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

2 Select Boundary 6 only.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domain 2 only.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

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

3 From the **Material symmetry** list, choose **Orthotropic**.

Roller 1

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

2 Select Boundary 5 only.

Symmetry Plane 1

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

2 Select Boundary 6 only.

MATERIALS

Porous Material 1 (pmat1)

Add the material parameters for the deformation. Note that the radial direction is the through-thickness direction and the vertical direction is the cross-machine direction.

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 **Homogenized Properties** section.

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Density	rho	`rho_s`	kg/m³	Basic
Biot-Willis coefficient	alphaB	`alpha_B`	1	Poroelastic material
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	`{nu_mz, nu_mc, nu_cz}`	1	Orthotropic
Shear modulus	{Gvector1, Gvector2, Gvector3}	`{G_mz, G_mc, G_cz}`	N/m²	Orthotropic
Young's modulus	{Evector1, Evector2, Evector3}	`{E_zd, E_md, E_cd}`	Pa	Orthotropic

MULTIPHYSICS

Unsaturated Poroelasticity 1 (unporo1)

Use the pore pressure computed by the Moisture Transport in Solids interface in order to have no deformation when exposed to the initial ambient conditions.

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Unsaturated Poroelasticity 1 (unporo1)**.

2 In the **Settings** window for **Unsaturated Poroelasticity**, locate the **Poroelastic Coupling Properties** section.

3 From the p_{ref} list, choose **Initial pore pressure (mts/pms1)**.

MESH 1

Mapped 1

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

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

Distribution 1

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

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

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

4 In the **Number of elements** text field, type 30.

5 Select Boundaries 6 and 7 only.

6 In the **Element ratio** text field, type 8.

7 From the **Growth rate** list, choose **Exponential**.

Distribution 2

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

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

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

4 In the **Number of elements** text field, type 30.

5 In the **Element ratio** text field, type 8.

6 Select Boundaries 5 and 8 only.

7 Select the **Reverse direction** check box.

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Boundaries Facing Ambient**.

Mapped 2

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

2 In the **Settings** window for **Mapped**, click **Build All**.

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 **d**.

4 In the **Output times** text field, type `range(0,t_r/20,t_r) range(t_r,1[d],t_end)`.

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

RESULTS

Use the **Configuration** node to select which time step to display in 1D Plot Group nodes.

Multiselect Solution 1

1 In the **Results** toolbar, click **Configurations** and choose **Multiselect Solution**.

2 In the **Settings** window for **Multiselect Solution**, locate the **Solution** section.

3 From the **Time selection** list, choose **Interpolated**.

4 In the **Times (d)** text field, type `1[d] 10[d] 30[d] 90[d]`.

Relative Humidity (mts)

1 In the **Model Builder** window, under **Results** click **Relative Humidity (mts)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Color Legend** section.

3 Select the **Show maximum and minimum values** check box.

Surface 2

1 Right-click **Relative Humidity (mts)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

2 Select Domain 1 only.

Deformation 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box. In the associated text field, type `1`.

4 In the **Relative Humidity (mts)** toolbar, click **Plot**.

Water Content, Cross Machine

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Solution parameters** list, choose **From configuration**.

4. In the **Label** text field, type `Water Content, Cross Machine`.
5. Click to expand the **Title** section. From the **Title type** list, choose **None**.
6. Locate the **Plot Settings** section.
7. Select the **y-axis label** check box. In the associated text field, type `Water content (1)`.
8. Locate the **Axis** section. Select the **Manual axis limits** check box.
9. In the **x minimum** text field, type `0`.
10. In the **x maximum** text field, type `H/2`.
11. In the **y minimum** text field, type `0.06`.
12. In the **y maximum** text field, type `0.13`.
13. Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1. Right-click **Water Content, Cross Machine** and choose **Line Graph**.
2. Select Boundary 5 only.
3. In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `mts.wc/rho_s/(1-por)`.
5. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6. In the **Expression** text field, type `CD`.
7. Click to expand the **Legends** section. Select the **Show legends** check box.
8. In the **Water Content, Cross Machine** toolbar, click **Plot**.

Water Content, Cross Machine

In the **Model Builder** window, right-click **Water Content, Cross Machine** and choose **Duplicate**.

Water Content, Through Thickness

1. In the **Model Builder** window, under **Results** click **Water Content, Cross Machine 1**.
2. In the **Settings** window for **1D Plot Group**, type `Water Content, Through Thickness` in the **Label** text field.
3. Locate the **Axis** section. In the **x minimum** text field, type `Ro/1.1`.
4. In the **x maximum** text field, type `Ro`.

Line Graph 1

1. In the **Model Builder** window, expand the **Water Content, Through Thickness** node, then click **Line Graph 1**.
2. In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 Click to select the **Activate Selection** toggle button.

4 Select Boundary 6 only.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type ZD.

6 In the **Water Content, Through Thickness** toolbar, click **Plot**.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1/Solution 1 (sol1)>Solid Mechanics>Displacement (solid)**.

4 Click **Add Plot** in the window toolbar.

RESULTS

Surface 1

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

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 From the **Unit** list, choose **mm**.

4 In the **Displacement (solid)** toolbar, click **Plot**.