Created in COMSOL Multiphysics 6.0

Condensation Detection in an Electronic Device with Transport and Diffusion

Introduction

This tutorial extends the Condensation Detection in an Electronic Device application by taking moist air transport and diffusion into account.

In the former application, the water vapor concentration is assumed to be homogeneous in the device. Here, you add a **Moisture Transport in Air** interface to study the water vapor transport and the condensation on the walls.

In addition, the amount of liquid water accumulated on the interior walls of the box due to condensation is evaluated.

Model Definition

This example is based on the one described in Condensation Detection in an Electronic Device.

A convection-diffusion equation is added to compute the vapor concentration, with a diffusion coefficient of $2.6 \cdot 10^{-5}\ \mathrm{m^2/s}$.

At the slit level, an open boundary condition is set with an upstream relative humidity ϕ_{ustr} and an upstream temperature T_{ustr} taken from ambient data.

The amount of liquid water c_{l} accumulated on the interior walls of the box by condensation is computed by solving the following equation:

$$M_{\mathrm{v}} \frac{\partial c_{\mathrm{l}}}{\partial t} = -g_{\mathrm{evap}}$$

where the evaporation flux g_{evap} is deduced from the saturation conditions on the walls surfaces:

- In supersaturation conditions, that is, $c_{\mathrm{v}} > c_{\mathrm{sat}}$, there is condensation on the surfaces, and the flux is negative (outgoing flux on the boundaries of the computational domain), equal to $M_{\mathrm{v}} K (c_{\mathrm{sat}} - c_{\mathrm{v}})$. The liquid concentration on the surface increases.
- In subsaturation conditions, that is, $c_{\mathrm{v}} < c_{\mathrm{sat}}$, and when there is some liquid on the surface, there is evaporation from the surfaces, and the flux is positive (ingoing flux on the boundaries of the computational domain), equal to $M_{\mathrm{v}} K (c_{\mathrm{sat}} - c_{\mathrm{v}})$. The liquid concentration on the surface decreases.
- In subsaturation conditions, that is, $c_{\mathrm{v}} < c_{\mathrm{sat}}$, and when there no liquid on the surface, the flux is null.

The evaporation rate K is set to $1\ \mathrm{m/s}$. A higher value does not make significant difference on the numerical results, while increasing the numerical stiffness of the model.

Results and Discussion

Figure 1 plots the maximum relative humidity for this study together with the previous results obtained in Condensation Detection in an Electronic Device.

The blue curve represents the maximum relative humidity without moisture transport; the red curve corresponds to the maximum relative humidity with varying moisture content and transport study; and the related saturation indicators are represented in green and cyan, respectively.

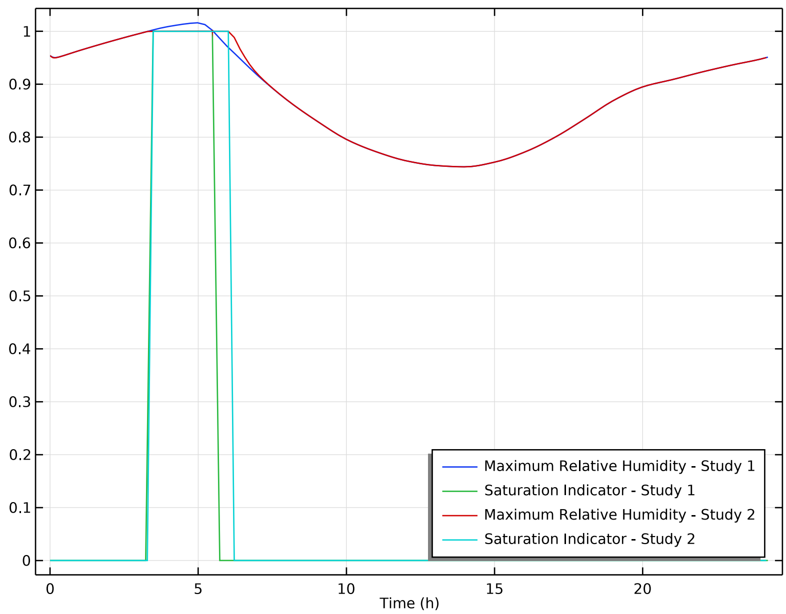

Figure 1: Maximum relative humidity of the two studies over time.

In both models, saturation occurs around 3 hours, this time corresponding with the time of the highest ambient relative humidity. When vapor diffusion is modeled as instantaneous as in the first study, the period of time in which saturation occurs is underestimated. The supersaturation conditions (relative humidity over 1) observed in the first study are not reproduced when the condensation on the walls is accounted for.

In addition, when the **Moisture Transport in Air** interface is used to model vapor transport, a more accurate spatial representation of the vapor field is obtained. Since the vapor saturation level depends on the local values of the concentration, temperature and pressure, this improves the condensation detection.

Shortly after 3 hours, the saturation conditions in the box induce condensation on its walls. The liquid water concentration begins to be positive on these surfaces, to reach its maximum at around 5 hours, then decrease again. Figure 2 shows the liquid water concentration at the left slit over time.

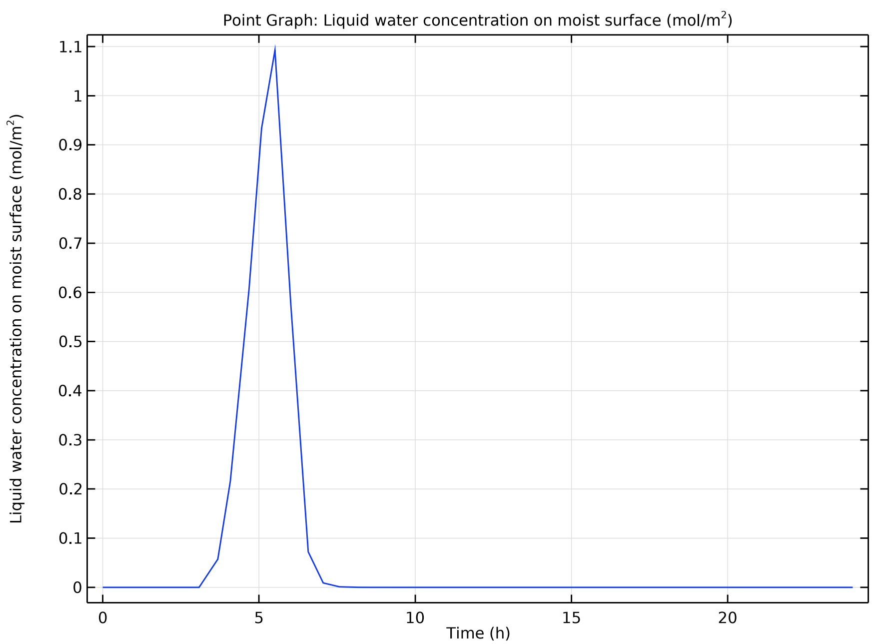

Figure 2: Liquid water concentration at the left slit over time.

Application Library path: `Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/condensation_electronic_device_transport_diffusion`

Modeling Instructions

ROOT

The second part of this simulation introduces transport and diffusion of water vapor. For this purpose, you add the **Moisture Transport in Air** interface to solve for the concentration.

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **Heat Transfer Module>Power Electronics and Electronic Cooling>condensation_electronic_device** in the tree.

3 Click **Open**.

ADD PHYSICS

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

2 Go to the **Add Physics** window.

3 In the tree, select **Chemical Species Transport>Moisture Transport>Moisture Transport in Air (mt)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

5 Click **Add to Component 1** in the window toolbar.

6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

MOISTURE TRANSPORT IN AIR (MT)

1 In the **Settings** window for **Moisture Transport in Air**, locate the **Domain Selection** section.

2 Click **Clear Selection**.

3 Select Domain 2 only.

MULTIPHYSICS

Heat and Moisture 1 (ham1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Heat and Moisture**.

2 In the **Model Builder** window, click **Heat and Moisture 1 (ham1)**.

3 Select Domain 2 only.

MOISTURE TRANSPORT IN AIR (MT)

Initial Values 1

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

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

3 From the $\phi_{w,0}$ list, choose **Ambient relative humidity (ampr1)**.

Open Boundary 1

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

2 Select Boundaries 3 and 22 only.

3 In the **Settings** window for **Open Boundary**, locate the **Upstream Properties** section.

4 From the T_{ustr} list, choose **Ambient temperature (ampr1)**.

5 From the $\phi_{w,ustr}$ list, choose **Ambient relative humidity (ampr1)**.

Add a **Moist Surface** feature on the interior walls of the box to account for evaporation and condensation on these surfaces, and evaluate the amount of liquid water accumulated over time.

Moist Surface 1

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

2 Select Boundaries 1, 2, 4–21, and 23 only.

3 In the **Settings** window for **Moist Surface**, locate the **Moist Surface Settings** section.

4 In the K text field, type 1.

MULTIPHYSICS

Moisture Flow 1 (mf1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Moisture Flow**.

Switch to the **Concentrated species** formulation. This adds in the **Moisture Flow** coupling node the option to account for Stefan flow at evaporation surfaces.

MOISTURE TRANSPORT IN AIR (MT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Moisture Transport in Air (mt)**.

2 In the **Settings** window for **Moisture Transport in Air**, locate the **Physical Model** section.

3 From the **Mixture type for moist air** list, choose **Concentrated species**.

MULTIPHYSICS

Moisture Flow 1 (mf1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Moisture Flow 1 (mf1)**.

2 In the **Settings** window for **Moisture Flow**, locate the **Moisture Transport at Walls** section.

3 Select the **Account for Stefan velocity at walls** check box.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Moist Air 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Heat Transfer in Solids and Fluids (ht)** node, then click **Moist Air 1**.

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

3 From the **Input quantity** list, choose **Concentration**.

4 Locate the **Model Input** section. From the *c* list, choose **Vapor concentration (mt)**.

This step couples the **Moisture Transport in Air** interface to the **Heat Transfer in Solids** interface.

In this second study, add probe curves to the first curves with the following steps.

DEFINITIONS

Domain Probe 1 (dom1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **Domain Probe 1 (dom1)**.

2 In the **Settings** window for **Domain Probe**, click to expand the **Table and Window Settings** section.

3 From the **Output table** list, choose **New table**.

Domain Probe 2 (dom2)

1 In the **Model Builder** window, click **Domain Probe 2 (dom2)**.

2 In the **Settings** window for **Domain Probe**, locate the **Table and Window Settings** section.

3 From the **Output table** list, choose **New table**.

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 **Add Study** 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 **h**.

3 Click **Range**.

4 In the **Range** dialog box, type `0.5` in the **Step** text field.

5 In the **Stop** text field, type `24`.

6 Click **Replace**.

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, locate the **General** section.

4 From the **Times to store** list, choose **Steps taken by solver closest to output times**.

5 Click to expand the **Time Stepping** section. From the **Maximum step constraint** list, choose **Constant**.

6 In the **Maximum step** text field, type `0.2`.

Because the temperature and pressure variations can be quick, forcing a reduced time step helps to capture all curve variations.

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

RESULTS

Ambient Temperature

The same default plots as before are generated automatically.

To reproduce the plot in Figure 1, edit the **Maximum Relative Humidity** plot group. Add the curves for the saturation indicator and maximum relative humidity for this new study.

Maximum Relative Humidity

1 In the **Model Builder** window, click **Maximum Relative Humidity**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Window Settings** section.

3 From the **Plot window** list, choose **Graphics**.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Maximum Relative Humidity** node, then click **Probe Table Graph 1**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 In the table, enter the following settings:

Legends
`Maximum Relative Humidity - Study 1`
`Saturation Indicator - Study 1`

Probe Table Graph 2

1 In the **Model Builder** window, click **Probe Table Graph 2**.

2 In the **Settings** window for **Table Graph**, locate the **Legends** section.

3 From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
`Maximum Relative Humidity - Study 2`

Probe Table Graph 3

1 In the **Model Builder** window, click **Probe Table Graph 3**.

2 In the **Settings** window for **Table Graph**, locate the **Legends** section.

3 From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
`Saturation Indicator - Study 2`

5 In the **Maximum Relative Humidity** toolbar, click **Plot**.

Create a plot showing the concentration of liquid water accumulated at the left slit over time, as in Figure 2.

Liquid water concentration over time

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

2 In the **Settings** window for **1D Plot Group**, type `Liquid water concentration over time` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Point Graph 1

1 Right-click **Liquid water concentration over time** and choose **Point Graph**.

2 Select Point 3 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `mt.cl_evap`.

5 In the **Liquid water concentration over time** toolbar, click **Plot**.

Mass Balance

Finally, follow the instructions below to check the overall mass balance over time.

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Mass Balance` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Moisture Transport in Air>Mass balance> mt.massBalance - Mass balance - kg/s**.

5 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Moisture Transport in Air>Mass balance> mt.dwcInt - Total accumulated moisture rate - kg/s**.

6 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Moisture Transport in Air>Mass balance> mt.ntfluxInt - Total net moisture rate - kg/s**.

7 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Moisture Transport in Air>Mass balance>mt.GInt - Total mass source - kg/s**.

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Moisture Transport in Air>Mass balance> Net mass flows, boundary features>mt.msu1.ntfluxInt - Total net moisture rate - kg/s**.

9 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Moisture Transport in Air>Mass balance> Net mass flows, boundary features>mt.open1.ntfluxInt - Total net moisture rate - kg/s**.

10 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
mt.msu1.ntfluxInt	kg/s	Total net moisture rate, evaporation/condensation
mt.open1.ntfluxInt	kg/s	Total net moisture rate, inlet/outlet

11 Click **=** **Evaluate**.

TABLE

1 Go to the **Table** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Mass Balance

1 In the **Model Builder** window, under **Results** click **1D Plot Group 15**.

2 In the **Settings** window for **1D Plot Group**, type `Mass Balance` 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**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 Locate the **Coloring and Style** section. In the **Width** text field, type 2.