Created in COMSOL Multiphysics 5.6

Condensation Detection in an Electronic Device

Introduction

Many systems, for example electronic devices, risk being damaged if exposed to condensation. Given an amount of moisture in the air, saturation occurs when the temperature decreases to reach the dew point, and condensation may occur on surfaces. Numerical simulations are useful for obtaining knowledge relevant for preventing the formation of condensation.

Changes in air properties are the primary cause of condensation in some systems. This example simulates the thermodynamical evolution of moist air in an electronic box with the aim of detecting whether saturation occurs when the external environment properties change. The model uses meteorological data for the air temperature, pressure, and relative humidity, measured at New York, JFK station. The property data correspond to average conditions of dry bulb temperature and high conditions of dew point temperature, observed on the 1st of June.

In this simulation, you assume the water vapor concentration to be homogeneous inside the box and equal to the external concentration. Also, the model setup neglects diffusion but considers the external concentration changes during the simulation.

Note: An extension of this application solves for an inhomogeneous concentration computed from the Moisture Transport in Air interface. It takes transport and diffusion of the water vapor into account, and evaluates the amount of liquid water accumulated on the walls over time, due to condensation and evaporation. See Condensation Detection in an Electronic Device with Transport and Diffusion.

Model Definition

A box with square cross section of side 5 cm is placed in a moist air environment. It contains a heated electronic component and two small slits (1 mm thick) located at the left and right sides. The simulation is in a 2D cross section of the box, which is supposed to be long enough in the orthogonal direction. It is made of aluminum and the electronic component is made of silicon. Figure 1 shows the model geometry.

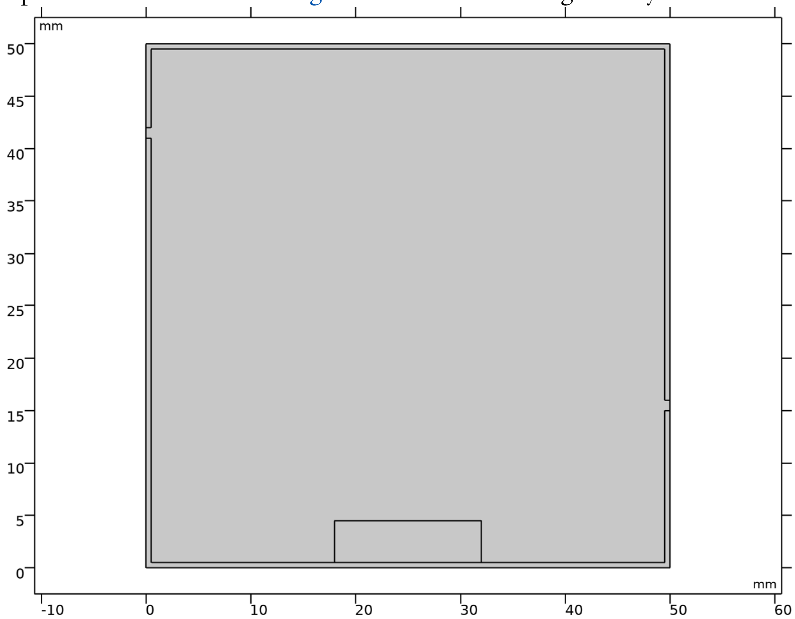

Figure 1: Geometry of the model.

The box is placed in a changing environment. This means that during the simulation, temperature and relative humidity change. Figure 2 shows the temperature and relative humidity as functions of time.

In this simulation, assume the moist air concentration inside the box to be equal to the external concentration.

Outside the box, you apply a convective cooling condition with a heat transfer coefficient h equal to 10 $W/(m^2 \cdot K)$ and a time-dependent external temperature equal to the ambient temperature. The central component produces a total power of 1 W during the simulation. At the slit boundaries, set a condition of open boundary to let external moist air freely enter or exit from the box.

The study computes a simulation over one day and the solution is stored every 30 minutes. The goal is to observe if some saturation appears to detect the risk of condensation.

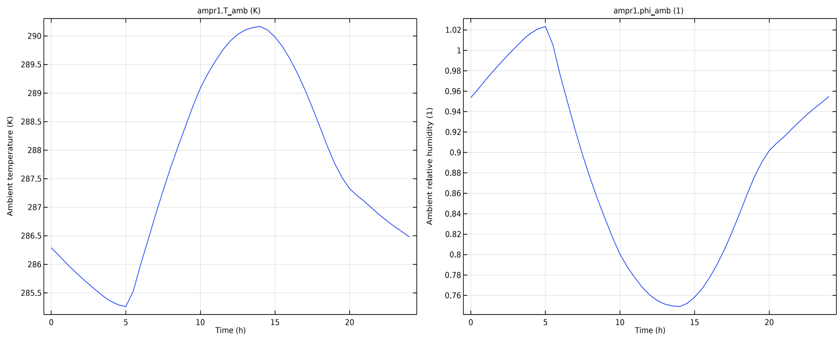

Figure 2: Temperature and relative humidity over the course of a day.

Results and Discussion

Figure 3 shows the temperature and relative humidity profiles at the final time step.

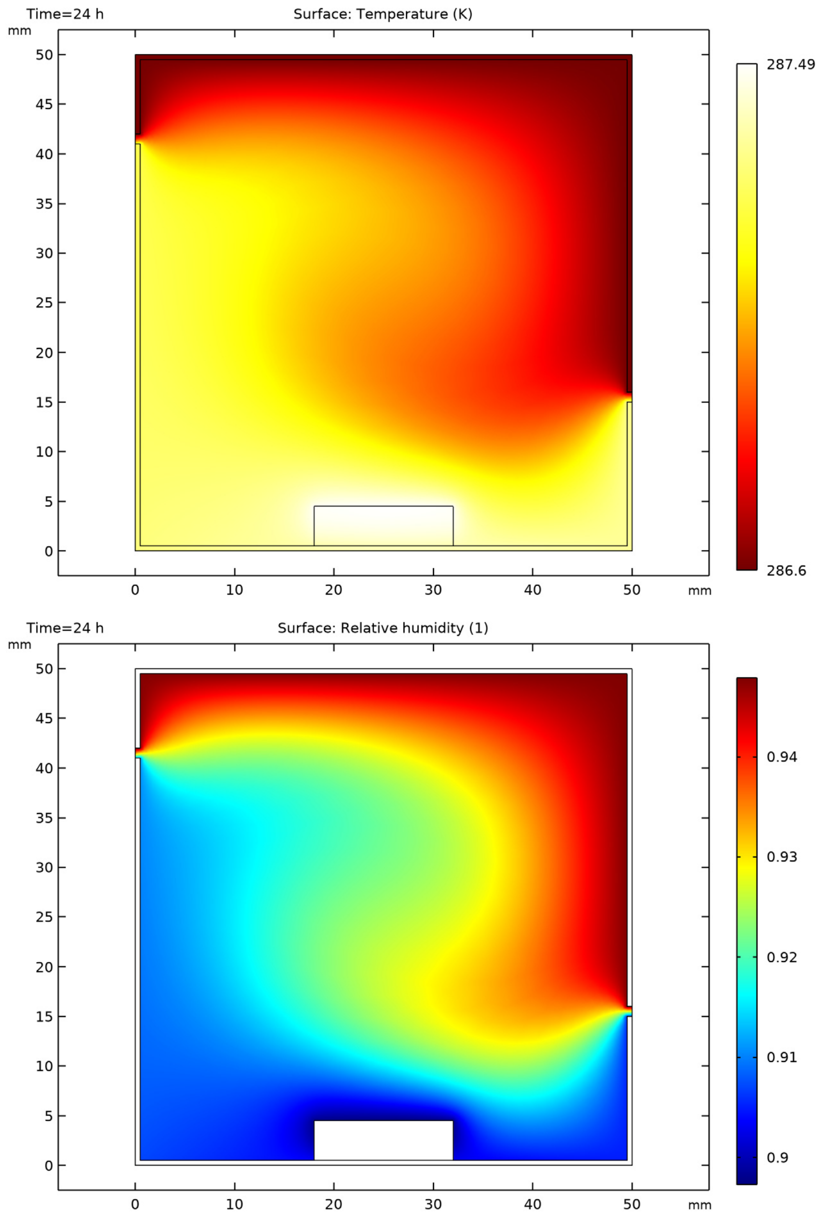

Figure 3: Temperature and relative humidity profiles after 24 hours.

While the temperature gradient is not very large, the power dissipated from the electronic component clearly influences the temperature field: it heats the surrounding air and the walls. Cold air enters through the slits by convection. In addition, the air inside the box is cooled by conduction through the walls. The relative humidity maximum is located where the temperature is the lowest but also where the water vapor concentration is the highest.

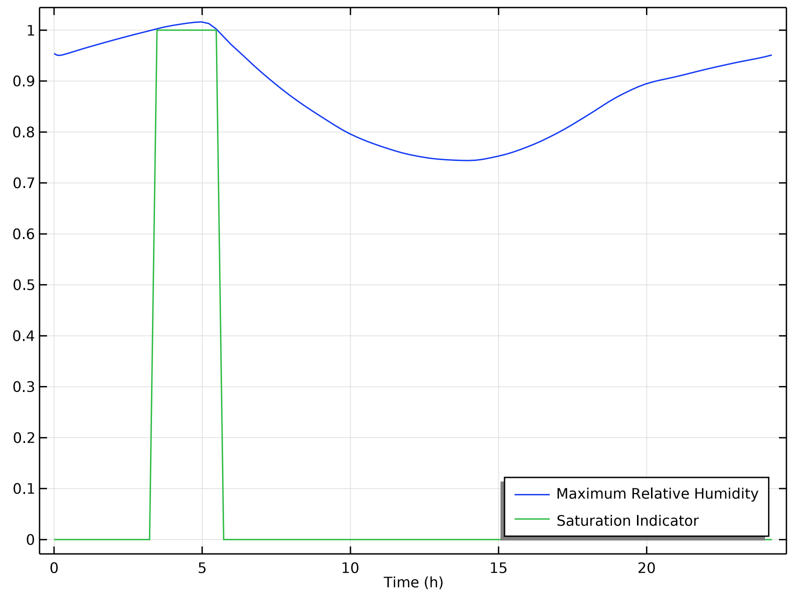

Figure 4: Maximum relative humidity over time inside the box.

Figure 4 represents the evolution of the maximum relative humidity inside the box over the simulation period. The saturation threshold, relative humidity equals 1, is reached around 3h, meaning that saturation occurs, with a risk of condensation on the box surfaces. A Boolean saturation indicator is inserted in order to distinguish the exact saturation period. The saturation indicator is set to 1 when saturation is detected (relative humidity greater or equals to 1) and to 0 otherwise.

Application Library path: `Heat_Transfer_Module/`
`Power_Electronics_and_Electronic_Cooling/`
`condensation_electronic_device`

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 **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Time Dependent**.
6. Click **Done**.

GEOMETRY 1

The geometry sequence for the model is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the Geometry Modeling Instructions section. Otherwise, insert the geometry sequence as follows:

1. In the **Geometry** toolbar, click **Insert Sequence**.
2. Browse to the model's Application Libraries folder and double-click the file `condensation_electronic_device_geom_sequence.mph`.
3. In the **Geometry** toolbar, click **Build All**.
4. Click the **Zoom Extents** button in the **Graphics** toolbar.

   You should now see the geometry shown in Figure 1.

MATERIALS

A material is only needed on the solid part as the fluid part is going to be defined at the feature level through the moist air functionality.

ADD MATERIAL

1. In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **Built-in>Aluminum**.
4. Click **Add to Component** in the window toolbar.
5. In the tree, select **Built-in>Silica glass**.

6 Click **Add to Component** in the window toolbar.

7 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Aluminum (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat1)**.

2 Select Domains 1 and 3 only.

Silica glass (mat2)

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

2 Select Domain 4 only.

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 2013)**.

4 Locate the **Time** section. Find the **Local time** subsection. In the table, enter the following settings:

Hour	Minute	Second
0	00	00

5 Locate the **Ambient Conditions** section. From the **Temperature** list, choose **Low**.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Moist Air 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Heat Transfer in Solids and Fluids (ht)** and choose **Specific Media>Moist Air**.

2 Select Domain 2 only.

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

4 From the ϕ_w list, choose **Ambient relative humidity (ampr1)**.

5 From the $T_{\phi w}$ list, choose **Ambient temperature (ampr1)**.

6 From the $p_{\phi w}$ list, choose **Ambient absolute pressure (ampr1)**.

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 From the T list, choose **Ambient temperature (ampr1)**.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2 Select Boundaries 1, 2, 5, 7, 21, and 23 only.
3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
4 Click the **Convective heat flux** button.
5 In the h text field, type 10.
6 From the T_{ext} list, choose **Ambient temperature (ampr1)**.

Heat Source 1

1 In the **Physics** toolbar, click **Domains** and choose **Heat Source**.
2 In the **Settings** window for **Heat Source**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Electronic Component**.
4 Locate the **Heat Source** section. Click the **Heat rate** button.
5 In the P_0 text field, type 1.

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)**.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
2 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Moist Air**.
4 Locate the **Physical Model** section. Select the **Include gravity** check box.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Fluid Properties 1**.
2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.

3 From the μ list, choose **Dynamic viscosity (ht/ma1)**.

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 p text field, type `ampr1.p_amb-spf.pref`.

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 **Boundary Condition** section.

4 In the f_0 text field, type `ampr1.p_amb-spf.pref`.

Then, two probes are defined in order to get the maximum relative humidity and the saturation indicator at the solver time steps.

DEFINITIONS

Domain Probe 1 (dom1)

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

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

3 Click **Clear Selection**.

4 From the **Selection** list, choose **Moist Air**.

5 Locate the **Probe Type** section. From the **Type** list, choose **Maximum**.

6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids and Fluids>Moist air>ht.phi - Relative humidity**.

Domain Probe 2 (dom2)

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

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

3 Click **Clear Selection**.

4 From the **Selection** list, choose **Moist Air**.

5 Locate the **Probe Type** section. From the **Type** list, choose **Maximum**.

6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids and Fluids>Moist air>ht.satInd - Saturation indicator**.

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 Click **Range**.
5 In the **Range** dialog box, type `0.5` in the **Step** text field.
6 In the **Stop** text field, type `24`.
7 Click **Replace**.
8 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
9 From the **Tolerance** list, choose **User controlled**.
10 In the **Relative tolerance** text field, type `0.0005`.

   Setting a lower relative tolerance avoids overshoots in the relative humidity solution at the beginning of the simulation.

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 From the **Maximum step constraint** list, choose **Constant**.
5 In the **Maximum step** text field, type `0.25`.

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

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

RESULTS

Temperature (ht)

This default plot represents the temperature profile at the last time step, as shown in the top panel of Figure 3.

Velocity (spf)

This default plot shows the velocity profile at the last time step.

Relative Humidity

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

Surface 1

1 In the **Relative Humidity** toolbar, 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)> Heat Transfer in Solids and Fluids>Moist air>ht.phi - Relative humidity**.

3 In the **Relative Humidity** toolbar, click **Plot**.

Compare with the relative humidity profile in the bottom panel of Figure 3.

Maximum Relative Humidity

Follow the steps below to reproduce the relative humidity evolution shown in Figure 4.

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 5**.

2 In the **Settings** window for **1D Plot Group**, type `Maximum Relative Humidity` in the **Label** text field.

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

4 In the table, enter the following settings:

Legends
`Maximum Relative Humidity`
`Saturation Indicator`

Maximum Relative Humidity

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

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

3 From the **Position** list, choose **Lower right**.

4 Click to expand the **Window Settings** section. From the **Plot window** list, choose **Graphics**.

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

Finally, plot the ambient temperature and relative humidity to reproduce Figure 2.

Ambient Temperature

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

2 In the **Settings** window for **1D Plot Group**, type `Ambient Temperature` in the **Label** text field.

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

4 In the **Title** text area, type `ampr1.T_amb (K)`.

Point Graph 1

1 Right-click **Ambient Temperature** and choose **Point Graph**.

2 Select Point 10 only.

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

4 In the **Expression** text field, type `ampr1.T_amb`.

5 In the **Ambient Temperature** toolbar, click **Plot**.

This figure should look like the left curve of Figure 2.

Ambient Relative Humidity

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

2 In the **Settings** window for **1D Plot Group**, type `Ambient Relative Humidity` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `ampr1.phi_amb (1)`.

Point Graph 1

1 Right-click **Ambient Relative Humidity** and choose **Point Graph**.

2 Select Point 10 only.

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

4 In the **Expression** text field, type `ampr1.phi_amb`.

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

This figure should look like the right curve of Figure 2.

Geometry Modeling Instructions

If you want to create the geometry yourself, follow these steps.

Geometry 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Square 1 (sq1)

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

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

3 In the **Side length** text field, type `50`.

Square 2 (sq2)

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

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

3 In the **Side length** text field, type `49`.

4 Locate the **Position** section. In the **x** text field, type `0.5`.

5 In the **y** text field, type `0.5`.

Electronic Component

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

2 In the **Settings** window for **Rectangle**, type `Electronic Component` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `14`.

4 In the **Height** text field, type `4`.

5 Locate the **Position** section. In the **x** text field, type `18`.

6 In the **y** text field, type `0.5`.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Rectangle 2 (r2)

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 `0.5`.

4 Locate the **Position** section. In the **y** text field, type `41`.

Rectangle 3 (r3)

1 Right-click **Rectangle 2 (r2)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 In the **x** text field, type `49.5`.

4 In the **y** text field, type `15`.

Union 1 (uni1)

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

2 Select the objects **r2**, **r3**, and **sq2** only.

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

4 Clear the **Keep interior boundaries** check box.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Moist Air

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Moist Air` in the **Label** text field.

3 On the object **fin**, select Domain 2 only.