Created in COMSOL Multiphysics 5.6

Evaporative Cooling of Water

Introduction

This tutorial shows how to simulate cooling of water including evaporative cooling. As an example, a beaker filled with water is used surrounded by an air domain. The airflow transports the water vapor which causes the liquid to cool down. The approach used here neglects volume change of the water inside the beaker. This is a reasonable assumption for problems where the considered time is short compared to the time needed to evaporate a noticeable amount of water.

Model Definition

The model geometry is shown in Figure 1. The size of the air domain is chosen such that increasing the domain would have no remarkable effect on the flow field around the beaker. Symmetry is used to reduce the model size.

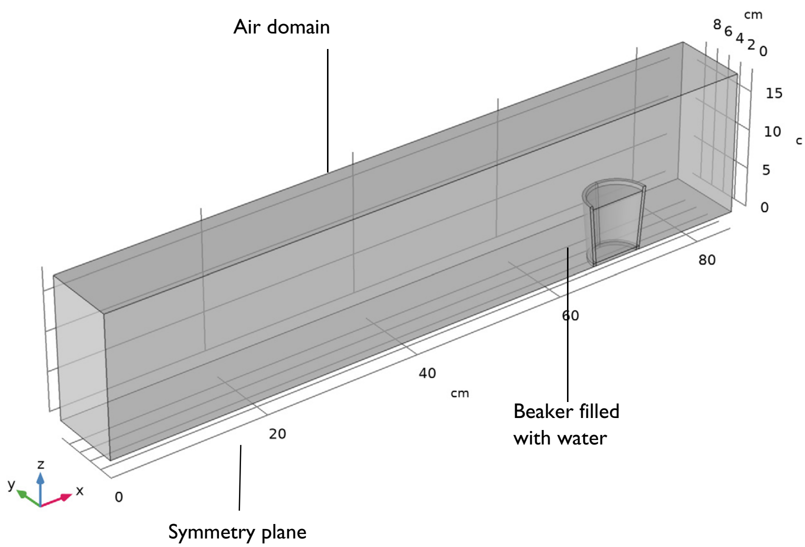


Figure 1: Model geometry, using symmetry.

The beaker is made of glass and contains hot water at 80°C. The air has an initial temperature of 20°C and enters the modeling domain with this temperature.

For modeling evaporative cooling, three effects must be taken into account: turbulent flow of the air around the beaker, heat transfer in all domains, and transport of water vapor in the air. This is a real multiphysics problem and this tutorial shows how to set it up.

TURBULENT FLOW

The airflow is modeled with the Turbulent Flow, Low Re k-ε interface, because the Reynolds number is about 1500 and turbulent effects must be considered. In addition, they must be taken into account in the transport equations correctly. With the Low Re k-ε turbulence model, the turbulence variables are solved in the whole domain down to the walls and thus provide accurate input values for the transport equations. Using the assumption that the velocity and pressure field are independent of the air temperature and moisture content makes it possible to calculate the turbulent flow field in advance and then use it as input for the heat transfer and species transport equations.

Note that because the mass contribution due to the evaporation is small at the water surface, a wall (no slip) condition is used on this boundary for the airflow computation.

HEAT TRANSFER

The heat transfer inside the beaker and water is due to conduction only. For the moist air, convection dominates the heat transfer and the turbulent flow field is required. The material properties are determined by the moist air theory.

During evaporation, latent heat is released from the water surface which cools down the water in addition to convective and conductive cooling by the surrounding. This additional heat flux depends on the amount of evaporated water. The latent heat source then is

$$q_{\mathrm{evap}} = L_{\mathrm{v}} g_{\mathrm{evap}} \tag{1}$$

The latent heat of vaporization L_{vap} is given in J/kg. The evaporative flux g_{evap} is discussed in the next section.

MOISTURE TRANSPORT

To obtain the correct amount of water which is evaporated from the beaker into the air, the Moisture Transport in Air interface is used. The initial relative humidity is 20%. At the water surface evaporation occurs. The evaporative flux at the surface is

$$g_{\mathrm{evap}} = K(c_{\mathrm{sat}} - c_{\mathrm{v}}) M_{\mathrm{v}} \tag{2}$$

with the evaporation rate K, the molar mass of water vapor M_{v}, the vapor concentration c_{v} and the saturation concentration c_{sat} which can be calculated from the correlation

$$c_{\mathrm{sat}} = \frac{p_{\mathrm{sat}}}{R_g T} \tag{3}$$

The transport equation again uses the turbulent flow field as input. Turbulence must also be considered for the diffusion coefficient, by adding the following turbulent diffusivity to the diffusion tensor:

$$\mathbf{D}_{\mathrm{T}} = \frac{\nu_{\mathrm{T}}}{\mathrm{Sc}_{\mathrm{T}}}\mathbf{I} \tag{4}$$

where ν_{T} is the turbulent kinematic viscosity, Sc_{T} is the turbulent Schmidt number, and $\mathbf{I}$ the unit matrix.

Results and Discussion

The image below shows the temperature field after 20 s with streamlines indicating the flow field.

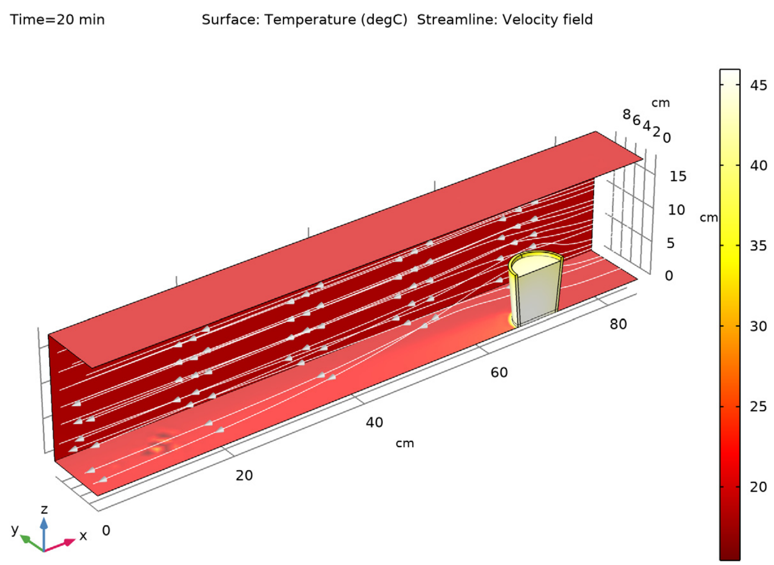


Figure 2: Temperature distribution after 20 min and streamlines indicating the flow field.

Due to convection, conduction, and evaporation the water cools down over time. As shown in Figure 3 the average temperature after 20 min is about 31°C.

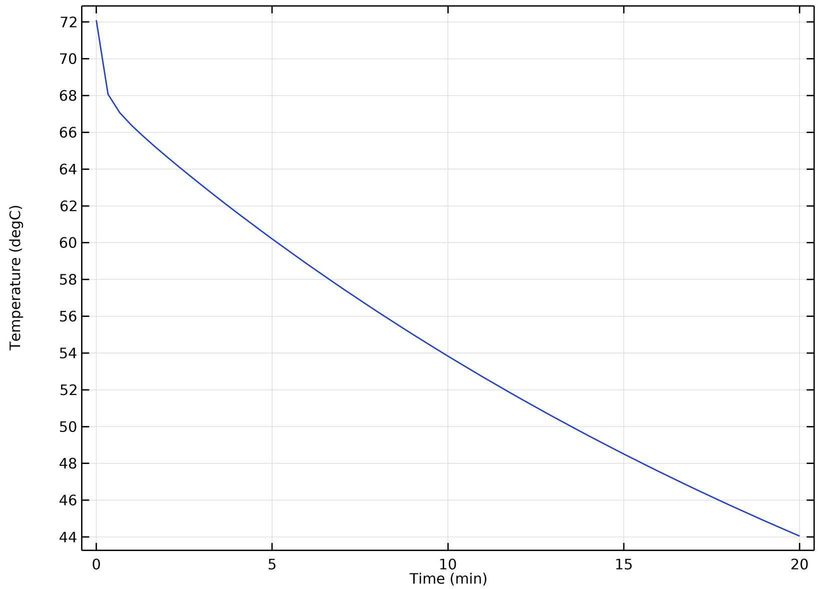


Figure 3: Average water temperature over time.

Figure 4 shows the concentration and relative humidity at the symmetry plane. Close to the water surface, the relative humidity is about 100% as expected. Behind the beaker the relative humidity can become even smaller than 20%. Due to the high temperature, air can absorb a higher amount of water.

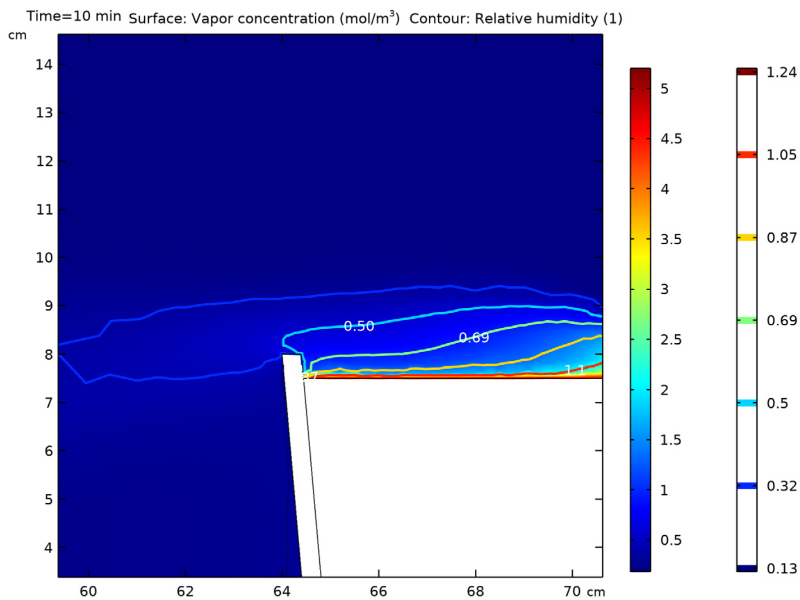


Figure 4: Concentration distribution and contour lines for the relative humidity.

The last study computes for a situation where the evaporation effects are neglected. Figure 5 shows the comparison between average water temperatures with and without evaporation to see the its importance in the cooling process. A difference of approximately 10 K can be observed.

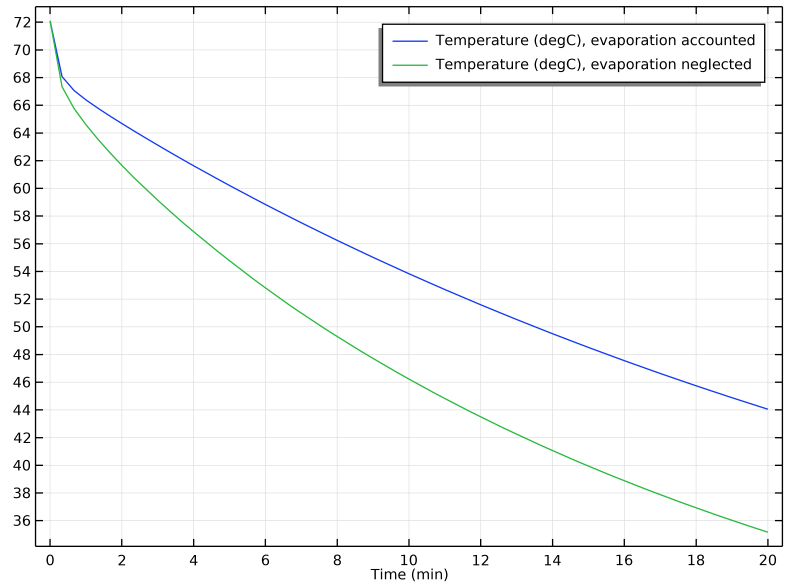


Figure 5: Average water temperature with and without evaporation accounted.

Application Library path: `Heat_Transfer_Module/Phase_Change/evaporative_cooling`

Modeling Instructions

The first step is to compute the turbulent flow field. After that, the resulting velocity field will be used to compute the transport of heat and moisture. To get an accurate velocity field for the turbulent transport equations, the Low-Reynolds k-ε turbulence model is used here.

From the **File** menu, choose **New**.

NEW

In the **New** window, click  **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, Low Re k-ε (spf)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary with Initialization**.
6 Click **Done**.

GEOMETRY 1

Load the geometry sequence from an existing MPH file.

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

   The flow calculation is done for the air domain only. For now, air is the only material you need.

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>Air**.
4 Click **Add to Component** in the window toolbar.
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

TURBULENT FLOW, LOW RE K-ε (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, Low Re k-ε (spf)**.
2 In the **Settings** window for **Turbulent Flow, Low Re k-ε**, locate the **Turbulence** section.
3 From the **Wall treatment** list, choose **Low Re**.

   This makes sure that the flow field is resolved down to the wall everywhere.

4 Locate the **Domain Selection** section. Click **Clear Selection**.

5 Select Domain 1 only.

Create a selection from this domain. It can be used later to create new selections or to assign physical properties.

6 Click **Create Selection**.

7 In the **Create Selection** dialog box, type `Air` in the **Selection name** text field.

8 Click **OK**.

Now, define the boundary conditions.

Inlet 1

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

2 Select Boundary 33 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `2`.

Open Boundary 1

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

2 Select Boundary 1 only.

Symmetry 1

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

2 Select Boundary 2 only.

MESH 1

Customize the mesh, so that it resolves both, the fluid flow and later the transport of heat and moisture properly. Use the Physics-controlled mesh as starting point.

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Extra coarse**.

4 Locate the **Mesh Settings** section. From the **Sequence type** list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

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

3 From the **Predefined** list, choose **Coarse**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Minimum element size** text field, type `1`.

Size 2

1 In the **Model Builder** window, click **Size 2**.

2 Remove boundaries 9, 12, and 29 from the list.

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

4 From the **Predefined** list, choose **Coarse**.

5 Click **Build Selected**.

Size 3

1 Right-click **Component 1 (comp1)>Mesh 1>Size 2** and choose **Duplicate**.

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

3 From the **Predefined** list, choose **Fine**.

4 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

5 Select Boundaries 9, 12, and 29 only.

6 Locate the **Element Size** section. Click the **Custom** button.

7 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

8 In the associated text field, type `0.2`.

Strong gradients appear near the water surface for both velocity, temperature and moisture content. Make the mesh elements smaller close the surface to improve accuracy.

Free Tetrahedral 1

1 In the **Model Builder** window, click **Free Tetrahedral 1**.

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

3 From the **Selection** list, choose **All domains**.

Boundary Layers 1

1 In the **Model Builder** window, click **Boundary Layers 1**.

2 In the **Settings** window for **Boundary Layers**, click to expand the **Corner Settings** section.

3 In the **Minimum angle for trimming** text field, type `350`.

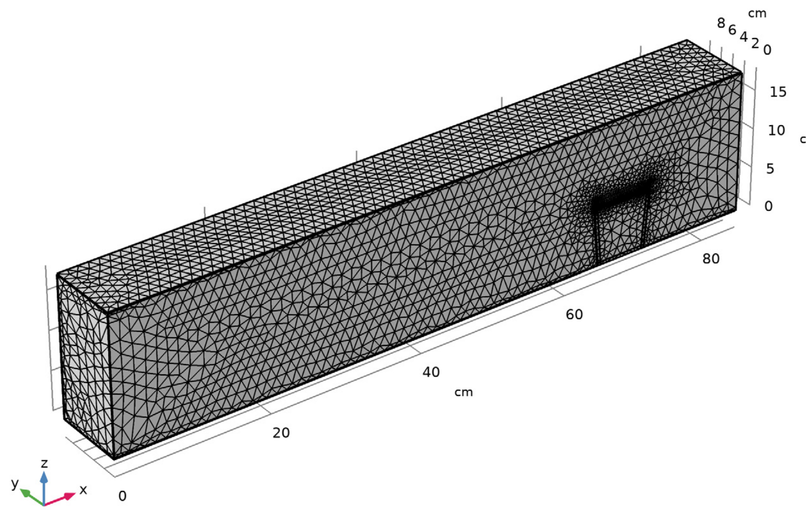
Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

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

3 In the **Number of boundary layers** text field, type 4.

4 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.



STUDY 1

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

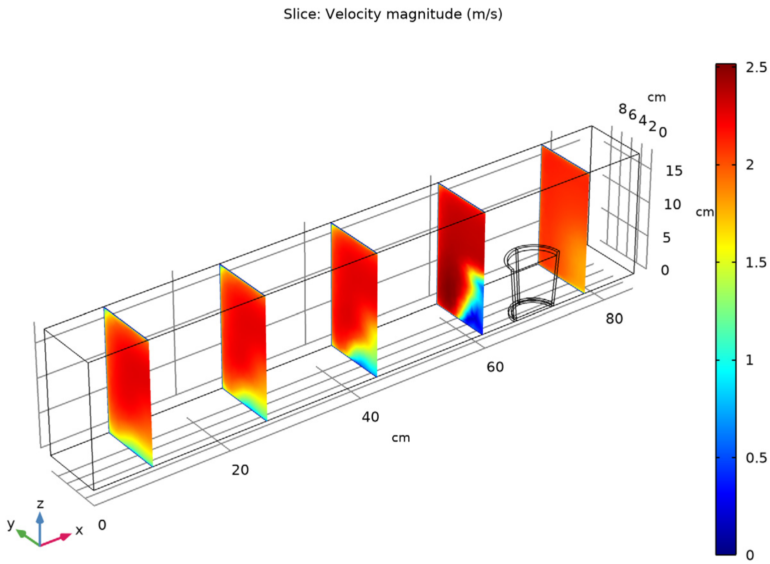
RESULTS

Velocity (spf)

1 Click the **Go to Default View** button in the **Graphics** toolbar.

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

The resulting velocity field is shown below:



With this velocity field, the transport equations can be computed. The **Heat Transfer in Moist Air** together with the **Moisture Transport in Air** interface are used to describe the transport of heat and moist air and the interaction of both processes.

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 **Heat Transfer>Heat and Moisture Transport>Moist Air**.

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.

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 In the table, enter the following settings:

Name	Expression	Value	Description
`phi0`	`0.2`	0.2	`Initial relative humidity`
`K`	`100[m/s]`	100 m/s	`Evaporation rate constant`

The evaporation rate is chosen so that the solution is not affected if the rate is further increased. This corresponds to assuming that vapor is in equilibrium with the liquid.

MATERIALS

Add the materials for heat transfer calculations in the cup and water.

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>Water, liquid**.

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

MATERIALS

Water, liquid (mat2)

1 Select Domain 3 only.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 Click **Create Selection**.

4 In the **Create Selection** dialog box, type `Water` in the **Selection name** text field.

5 Click **OK**.

With this selection and the one for the air domain, it is easy to create the selection for the glass body.

DEFINITIONS

Glass

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

2 Right-click **Definitions** and choose **Selections>Complement**.

3 In the **Settings** window for **Complement**, type `Glass` in the **Label** text field.

4 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

5 In the **Add** dialog box, in the **Selections to invert** list, choose **Air** and **Water**.

6 Click **OK**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-in>Glass (quartz)**.

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

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

MATERIALS

Glass (quartz) (mat3)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Glass**.

HEAT TRANSFER IN MOIST AIR (HT)

Add a **Fluid** node for the water domain. To save computational time, the velocity field driven by natural convection is not computed. Instead, an increased thermal conductivity determined by built-in Nusselt number correlations is defined in the next steps to compensate the missing convective heat flux.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Moist Air (ht)**.

Fluid 1

1 In the **Physics** toolbar, click **Domains** and choose **Fluid**.

2 Select Domain 3 only.

Convectively Enhanced Conductivity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Convectively Enhanced Conductivity**.

2 In the **Settings** window for **Convectively Enhanced Conductivity**, locate the **Convectively Enhanced Conductivity** section.

3 From the **Nusselt number correlation** list, choose **Vertical rectangular cavity**.

4 In the H text field, type `8[cm]`.

5 In the L text field, type `3.5[cm]`.

Then, add a **Solid** node for the glass domain.

Solid 1

1. In the **Physics** toolbar, click **Domains** and choose **Solid**.
2. In the **Settings** window for **Solid**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Glass**.

The air enters the domain at room temperature. At the outlet, the heat is transported away by convection.

Inflow 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.
2. Select Boundary 33 only.

Open Boundary 1

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

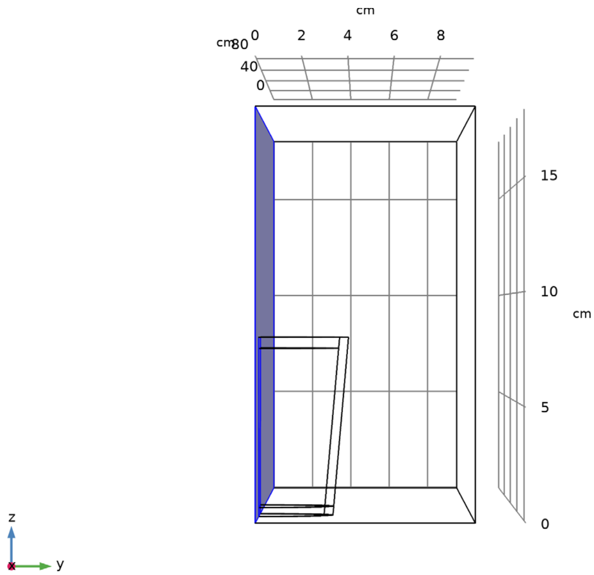
Symmetry 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2. Click the **Go to YZ View** button in the **Graphics** toolbar.

   With this tool, draw a box around all symmetry boundaries, which corresponds to:

3 Select Boundaries 2, 6, 11, 13, 18, 31, and 32 only.

You should see the following in your Graphics window:



The fluid in the beaker has an initial temperature of 80°C.

Initial Values 2

1 In the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 Select Domain 3 only.

3 In the **Settings** window for **Initial Values**, locate the **Domain Selection** section.

4 From the **Selection** list, choose **Water**.

5 Locate the **Initial Values** section. In the T text field, type `80[degC]`.

Set up the **Moisture Transport in Air** interface.

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 **Domain Selection** section.

3 From the **Selection** list, choose **Air**.

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. In the $\phi_{w,0}$ text field, type `phi0`.

Inflow 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.
2. Select Boundary 33 only.
3. In the **Settings** window for **Inflow**, locate the **Upstream Properties** section.
4. In the $\phi_{w,ustr}$ text field, type `phi0`.

Open Boundary 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Open Boundary**, locate the **Upstream Properties** section.
4. In the $\phi_{w,ustr}$ text field, type `phi0`.

Symmetry 1

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

Wet Surface 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Wet Surface**.
2. Select Boundary 12 only.
3. In the **Settings** window for **Wet Surface**, locate the **Wet Surface Settings** section.
4. In the K text field, type `K`.

Now, set up the multiphysics couplings for moisture and heat transport by the airflow. Start with a fictive model where latent heat source due to evaporation is neglected.

MULTIPHYSICS

Heat and Moisture 1 (ham1)

1. In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Heat and Moisture 1 (ham1)**.
2. In the **Settings** window for **Heat and Moisture**, locate the **Latent Heat** section.
3. Clear the **Include latent heat source on surfaces** check box.

Nonisothermal Flow 1 (nitf1)

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

2 In the **Settings** window for **Nonisothermal Flow**, locate the **Material Properties** section.

3 Select the **Boussinesq approximation** check box.

Couple the flow and pressure field.

Moisture Flow 1 (mf1)

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

To keep **Study 1** in its original state, deselect these last two multiphysics coupling features from the study tables.

STUDY 1

Step 1: Wall Distance Initialization

1 In the **Model Builder** window, under **Study 1** click **Step 1: Wall Distance Initialization**.

2 In the **Settings** window for **Wall Distance Initialization**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check boxes for **Nonisothermal Flow 1 (nitf1)** and **Moisture Flow 1 (mf1)**.

Step 2: Stationary

1 In the **Model Builder** window, click **Step 2: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check boxes for **Nonisothermal Flow 1 (nitf1)** and **Moisture Flow 1 (mf1)**.

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Turbulent Flow, Low Re k-ε (spf)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

5 Click **Add Study** in the window toolbar.

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

STUDY 2 : NO LATENT HEAT SOURCE

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2 : no latent heat source` in the **Label** text field.

Step 1: Time Dependent

Due to the **Wet Surface** boundary condition, the time-dependent simulation of heat- and moisture transport is very sensitive to the choice of the time dependent solver settings. Tune the solver, by restricting the time step size.

1 In the **Model Builder** window, under **Study 2 : no latent heat source** 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 **min**.

4 In the **Output times** text field, type `range(0,20[s],20)`.

5 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

6 From the **Method** list, choose **Solution**.

7 From the **Study** list, choose **Study 1, Stationary**.

Because you do not solve for the flow field again, but want to use the results from the first study, you have to tell the time-dependent study that the results from the stationary study should be used.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** 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 `1`.

6 Right-click **Time-Dependent Solver 1** and choose **Fully Coupled**.

7 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

8 From the **Jacobian update** list, choose **On every iteration**.

Step 1: Time Dependent

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

RESULTS

Temperature (ht)

Create the plot shown in Figure 2.

Surface 2

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

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

3 From the **Dataset** list, choose **Study 2 : no latent heat source/Solution 3 (sol3)**.

4 Locate the **Selection** section. From the **Selection** list, choose **All boundaries**.

5 Remove boundaries 1, 2 and 33 from the list.

Temperature (ht)

1 In the **Model Builder** window, click **Temperature (ht)**.

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

3 From the **Dataset** list, choose **Surface 2**.

Surface

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

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

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

Temperature (ht)

In the **Model Builder** window, click **Temperature (ht)**.

Streamline 1

1 In the **Temperature (ht)** toolbar, click **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2 : no latent heat source/Solution 3 (sol3)**.

4 From the **Solution parameters** list, choose **From parent**.

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **White**.

6 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **On selected boundaries**.

7 Locate the **Selection** section. Select the **Activate Selection** toggle button.

8 Select Boundary 33 only.

9 Locate the **Coloring and Style** section. From the **Type** list, choose **Arrow**.

10 Select the **Number of arrows** check box.

11 In the associated text field, type `80`.

12 In the **Temperature (ht)** toolbar, click **Plot**.

13 Click the **Go to Default View** button in the **Graphics** toolbar.

To visualize the moisture distribution as in Figure 4, follow the next steps.

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.

3 From the **Plane** list, choose **xz-planes**.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 2 : no latent heat source/ Solution 3 (sol3)**.

Moisture Concentration and Relative Humidity

1 In the **Results** toolbar, click **2D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Plane 1**.

4 From the **Time (min)** list, choose **10**.

Surface 1

1 In the **Moisture Concentration and 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)> Moisture Transport in Air>mt.cv - Vapor concentration - mol/m³**.

Moisture Concentration and Relative Humidity

In the **Model Builder** window, click **Moisture Concentration and Relative Humidity**.

Contour 1

1 In the **Moisture Concentration and Relative Humidity** toolbar, click **Contour**.

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

3 In the **Expression** text field, type `mt.phi`.

4 Locate the **Levels** section. In the **Total levels** text field, type `7`.

5 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Tube**.

6 Select the **Radius scale factor** check box.

7 In the associated text field, type `0.025`.

8 Select the **Level labels** check box.

9 In the **Precision** text field, type `2`.

10 From the **Label color** list, choose **White**.

11 In the **Moisture Concentration and Relative Humidity** toolbar, click **Plot**.

Use the **Zoom Box** button in the **Graphics** window to get a better view of the contour lines.

The relative humidity decreases quickly with the distance to the surface. Due to the high temperature behind the beaker, the relative humidity becomes even lower than 20%.

It is interesting to see how the average temperature decreases with time.

Average Water Temperature

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Average> Volume Average**.

2 In the **Settings** window for **Volume Average**, type `Average Water Temperature` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 : no latent heat source/ Solution 3 (sol3)**.

4 Select Domain 3 only.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Moist Air>Temperature>T - Temperature - K**.

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

Expression	Unit	Description
T	degC	Temperature

7 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

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

RESULTS

Average Water Temperature over Time

1. In the **Model Builder** window, under **Results** click **1D Plot Group 8**.
2. In the **Settings** window for **1D Plot Group**, type `Average Water Temperature over Time` in the **Label** text field.

Finally, compute how much water is evaporated in the air.

Amount of Evaporated Water

1. In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Integration> Surface Integration**.
2. In the **Settings** window for **Surface Integration**, type `Amount of Evaporated Water` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Study 2 : no latent heat source/ Solution 3 (sol3)**.
4. Select Boundary 12 only.
5. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`2*mt.ntflux`	`kg/s`	

6. Locate the **Data Series Operation** section. From the **Operation** list, choose **Integral**.
7. Click next to **Evaluate**, then choose **New Table**.

TABLE

1. Go to the **Table** window.

   The factor 2 in the expression is based on the use of a symmetry condition. Within 20 minutes, about 11.2 g of water have been evaporated.

MULTIPHYSICS

Repeat the previous steps with a third study that takes latent heat source due to evaporation into account. A comparison with the results returned by Study 2 will then highlight and quantify the cooling effects of evaporation.

Heat and Moisture 2 (ham2)

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

Heat and Moisture 1 (ham1)

1 In the **Model Builder** window, right-click **Heat and Moisture 1 (ham1)** and choose **Disable**.

To keep **Study 1** and **Study 2** in their original states, disable the **Heat and Moisture** multiphysics coupling feature from their physics trees.

STUDY 1

Step 1: Wall Distance Initialization

1 In the **Model Builder** window, under **Study 1** click **Step 1: Wall Distance Initialization**.

2 In the **Settings** window for **Wall Distance Initialization**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Heat and Moisture 2 (ham2)**.

Step 2: Stationary

1 In the **Model Builder** window, click **Step 2: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Heat and Moisture 2 (ham2)**.

STUDY 2 : NO LATENT HEAT SOURCE

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2 : no latent heat source** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Heat and Moisture 2 (ham2)**.

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Turbulent Flow, Low Re k-ε (spf)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

5 Click **Add Study** in the window toolbar.

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

STUDY 3: LATENT HEAT SOURCE

1. In the **Model Builder** window, click **Study 3**.
2. In the **Settings** window for **Study**, type `Study 3 : latent heat source` in the **Label** text field.

Step 1: Time Dependent

1. In the **Model Builder** window, 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 **min**.
4. In the **Output times** text field, type `range(0,20[s],20)`.
5. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
6. From the **Method** list, choose **Solution**.
7. From the **Study** list, choose **Study 1, Stationary**.

Solution 4 (sol4)

1. In the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Time-Dependent Solver 1**.
3. In the **Settings** window for **Time-Dependent Solver**, locate the **Time Stepping** section.
4. From the **Maximum step constraint** list, choose **Constant**.
5. In the **Maximum step** text field, type `1`.
6. Right-click **Time-Dependent Solver 1** and choose **Fully Coupled**.
7. In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.
8. From the **Jacobian update** list, choose **On every iteration**.

Step 1: Time Dependent

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

RESULTS

Temperature (ht) 1

Compute the average temperature of water when evaporation is neglected.

Average Water Temperature 1

1 In the **Model Builder** window, right-click **Average Water Temperature** and choose **Duplicate**.

2 In the **Settings** window for **Volume Average**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 3 : latent heat source/Solution 4 (sol4)**.

4 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

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

RESULTS

Table Graph 1

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Results>Average Water Temperature over Time>Table Graph 1**.

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

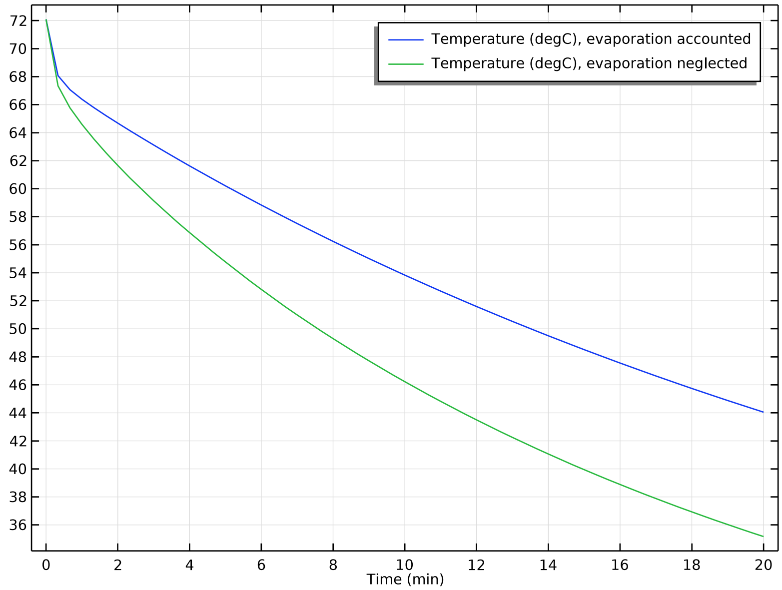
3 Select the **Show legends** check box.

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

5 In the table, enter the following settings:

Legends
`Temperature (degC), evaporation accounted`
`Temperature (degC), evaporation neglected`

6 In the **Average Water Temperature over Time** toolbar, click **Plot**.



In this model, evaporation accounts for a decrease of about 10 K at the end of the simulation (see Figure 5).