Evaporation of Ethanol and Water from a Wine Glass

Introduction

Wine shows are events that involve trying and judging different wines. When this is performed formally, the samples that are not assessed immediately are covered to avoid exposure to air. The air exposure would lead to evaporation of, for example, ethanol from the wine, which would affect the tasting experience. Occasionally, the event may involve more than 30 different samples of wine, where the wine is poured prior to judging. If the samples are uncovered, the latter ones will have been exposed for air for a longer time than the first ones. Inspired by the paper Wollan and others (Ref. 1), this model simulates the evaporation and transport of ethanol and water from a wine glass into an ambient air domain. Four different compositions in terms of the alcohol by volume are solved for: 0.01, 0.15, 0.4, and 0.99. Evaporation of multiple species from a nonideal liquid mixture is modeled using the Vapor–Liquid Interface feature. Evaporation at the liquid surface induces free convection in the surrounding vapor phase, both due to the change in composition and due to the heat of vaporization. The model is set up in a 2D axially symmetric model using coupled Laminar Flow, Transport of Concentrated Species in Vapor, and Heat Transfer in Fluids interfaces. Accurate thermodynamic data are provided by the Thermodynamics functionality.

Model Definition

The model geometry consists of a partially filled wine glass, placed on a table, and surrounded by a gas phase domain, which is initially filled with air. The relative humidity in the air domain is set to 30%, a value typical of indoor environments, and the temperature to 23°C. The regular geometry of the glass allows for a 2D axially symmetric geometry to be used (see Figure 1). The specific chemical composition of different wines varies, but in this model, it is assumed that only ethanol and water are present in the liquid. Evaporation of both species is accounted for by computing the equilibrium vapor phase mole fractions, and applying them at the liquid surface. The Reacting Flow multiphysics interface is used to solve for coupled fluid flow, mass transport, and heat transfer in the vapor region inside of the glass and around it. Heat transfer by conduction is solved for in the glass and the table. Assuming that the influence of convection in the liquid is small, it is assumed stationary and subjected to heat transfer by conduction only. In order to study how the alcohol content influences the evaporation, four different liquid compositions characterized by the alcohol by volume (abv): 0.01, 0.15, 0.4, and 0.99, are solved for.

An abv of 0.01 implies that out of the total liquid volume, 1% is ethanol and the 99% is water.

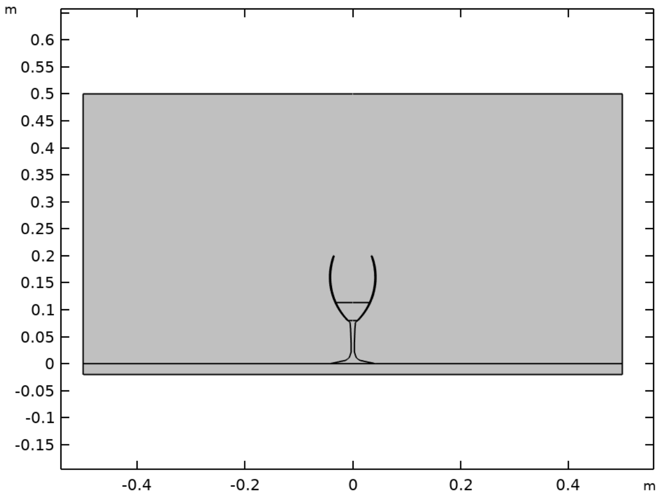


Figure 1: The model geometry, a wine glass placed on a table in a domain initially filled with air at a relative humidity of 30%.

VAPOR–LIQUID EQUILIBRIUM

To model the evaporation of ethanol and water from the glass of wine, the vapor-liquid equilibrium at the surface of the beverage must be described. The condition of thermodynamic equilibrium for a species i is given by

$$f_i^{\mathrm{L}}(T, p, \omega_{l,i}) = f_i^{\mathrm{V}}(T, p, \omega_{v,i}) \tag{1}$$

where f_i^{L} is the liquid phase fugacity and f_i^{V} is vapor phase fugacity. The fugacities in each phase depend on the temperature and pressure, T and p, as well as the mass fraction in the liquid phase, $\omega_{l,i}$, and the mass fraction in the vapor phase, $\omega_{v,i}$, respectively.

The thermodynamic functions for fugacity f_i $(=f_i^{\mathrm{L}}=f_i^{V})$ will be created automatically when all species are coupled to a Thermodynamics system. With the assumption of f_i being equal to $p_{\mathrm{eq},i}$, the molar concentration ($c_{\mathrm{eq},i}$) can be expressed as

$$c_{\mathrm{eq},i} = \frac{p_{\mathrm{eq},i}}{RT} \tag{2}$$

where R is the ideal gas constant, 8.314 J/(mol·K).

The mass fraction at the vapor–liquid surface at equilibrium is

$$\omega_{0,i} = \frac{p_{\mathrm{eq},i} M_i}{\rho} \tag{3}$$

This is the boundary condition at the vapor–liquid surface under the Transport of Concentrated Species in Vapor.

MODELING EVAPORATION

In the model, the mass fraction of ethanol and water vapor are prescribed at the vapor-liquid interface using Equation 3. This sets up diffusive transport of each species to or from the surface. The ethanol concentration in the vapor is initially zero implying that ethanol is transported from the surface into the vapor phase. To account for the assumption that the liquid surface does not move due to the evaporation, the fluid velocity normal to the surface is defined from the Stefan velocity

$$\rho u_{\mathrm{s}} = \mathbf{n} \cdot \sum_i (\mathbf{j}_i + \rho u_{\mathrm{s}} \omega_i \mathbf{n}) \tag{4}$$

Here, the sum on the right-hand side contains the total mass flux of all species for which the mass fraction is specified at the surface. The Stefan velocity implies that the mass flux of the species which is not controlled, nitrogen, is zero across the liquid surface. Since the vapor composition changes due to evaporation, the vapor phase is also subjected to natural convection. When the density changes due to evaporation, the vapor phase in the glass will start moving due to buoyancy.

The heat of evaporation at the liquid surface is

$$Q_{\mathrm{b}} = N_{\mathrm{e}} \Delta H_{\mathrm{vap,e}} + N_{\mathrm{w}} \Delta H_{\mathrm{vap,w}} \tag{5}$$

Here N is the normal mass flux across the liquid surface, ΔH_{vap} is the species heat of vaporization (J/kg). The Q_{b} is contributed from both the ethanol (index 'e') and water (index 'w'). The heat of evaporation can be picked up by a Boundary Heat Source feature under the Heat Transfer in Fluids interface.

The evaporation causes a heat loss at the liquid surface. The density in the vapor increases due to the lower temperature, which in turn induces or counteracts free convection.

Results and Discussion

For each of the four alcohol cases, the model is solved for 900 s of evaporation. In Figure 2, a composite plot showing the resulting velocity, temperature, and mass fractions of the evaporated species at the final time step for the case of abv = 0.01, is seen. It is evident from the velocity field that the vapor produced at the liquid surface is lighter than the surrounding air. The reason for this is that mostly water vapor is evaporated. As a consequence the vapor rises straight up from the glass. It can also be noted from the temperature field that the evaporation causes a temperature reduction close the liquid surface. Due to conduction in the glass, this also cools the ambient air on the outside of the glass, produces a downward motion in the air around the glass.

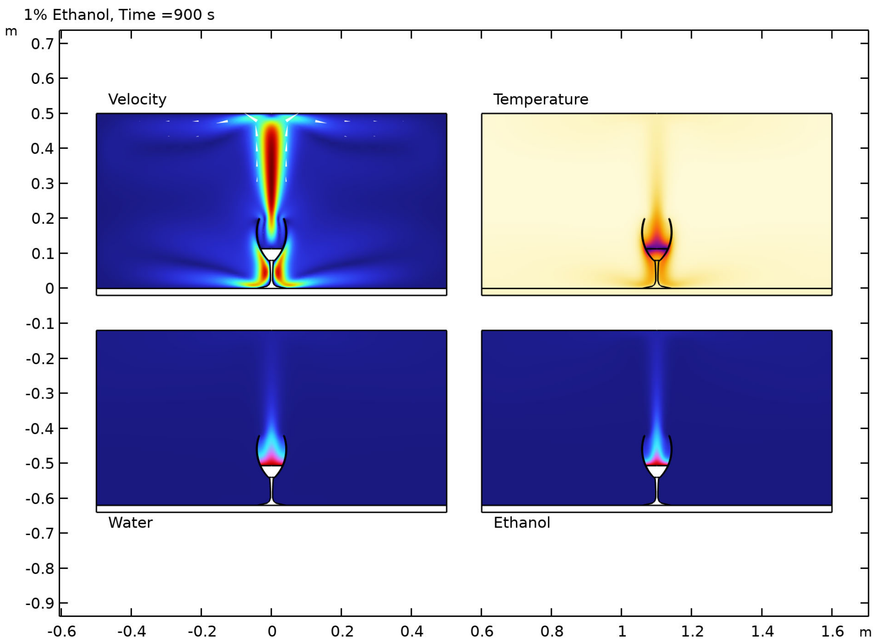


Figure 2: Velocity, temperature, and vapor phase mass fractions at t = 900 s for abv = 0.01.

The same type of figure for abv = 0.15, corresponding to an alcohol content similar to that of wine, is seen in Figure 3. In this case, a significantly higher ethanol concentration in the vapor is produced. Ethanol vapor has a higher density than air and the vapor inside the glass becomes heavier than the surrounding air and does not rise. Instead, the glass fills up with ethanol and water vapor from the liquid surface and up. Once the ethanol vapor reaches the brim of the glass it passes over and travels down on the outside of the glass.

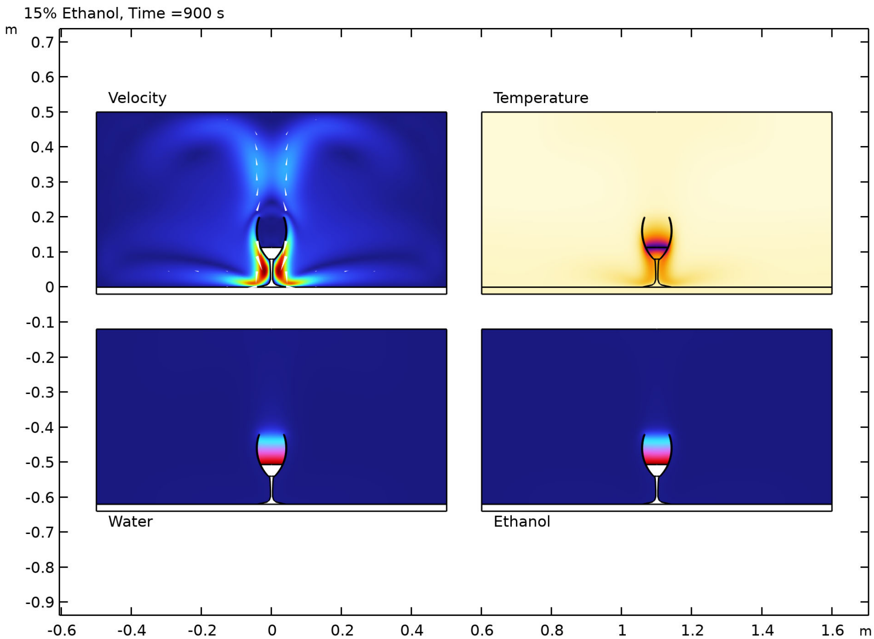


Figure 3: Velocity, temperature, and vapor phase mass fractions at t = 900 s for abv = 0.15.

Figure 4 shows the vapor velocity in the last time step for all four compositions computed. The development noted above continues also for the two higher alcohol content cases.

Higher concentrations of ethanol in the vapor phase leads to higher velocities in the vapor as it passes over the brim and down along the glass.

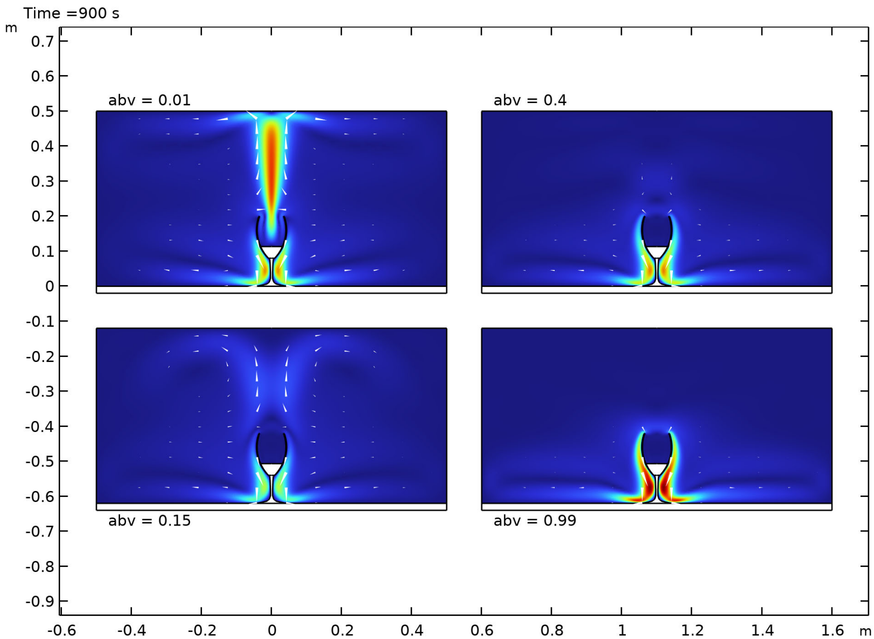


Figure 4: Vapor phase velocity at t = 900 s for all compositions.

Figure 5 shows the vapor mass flux, and accumulated mass, of water in the vapor phase for each abv. The vapor mass flux changes rapidly in the start but reaches an approximately linear development after around 200 s. The highest mass flux is seen for abv = 0.01. This case was noted above to be characterized by a rising convective stream due to buoyancy, which aids the transports vapor out of the glass. The mass flux for abv = 0.15 and abv = 0.4 are similar. But for the highest alcohol content, abv = 0.99, the vapor mass flux of water is negative throughout the entire time span. This implies that the water content in the air, due the relative humidity, is higher than the equilibrium vapor concentration at the liquid surface, leading to condensation of water into the liquid.

The vapor mass flux and accumulated mass ethanol is shown in Figure 6. The mass flux to the vapor phase is seen to strictly increase with the alcohol content in the liquid. This is in line with the fact that no ethanol is present in the air prior to the evaporation. It can also be noted that the accumulated mass of ethanol transported to the vapor is higher than that of water for all but the lowest alcohol content case.

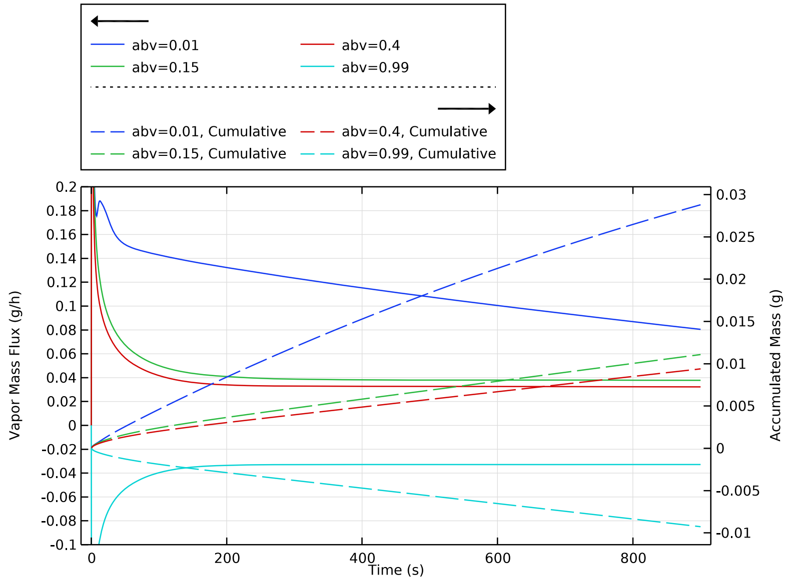


Figure 5: Vapor mass flux (solid lines) and accumulated mass (dashed lines) of water.

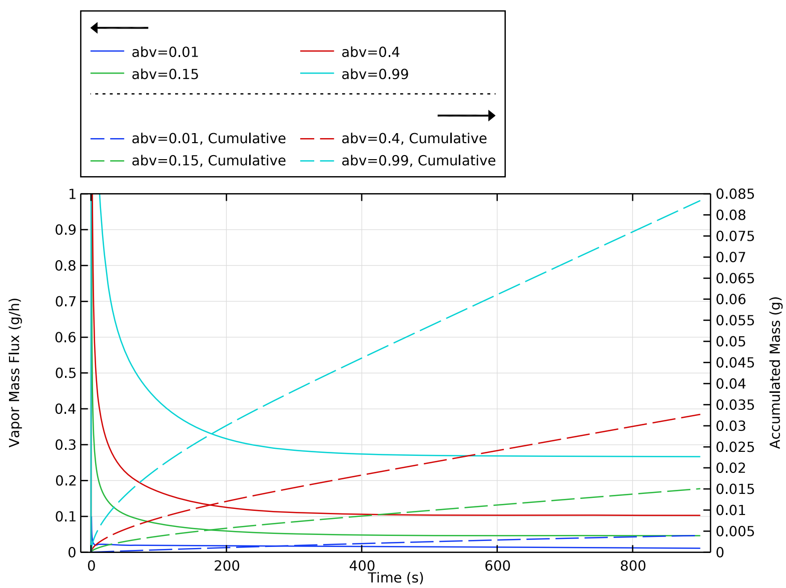


Figure 6: Vapor mass flux (solid lines) and accumulated mass (dashed lines) of ethanol.

The temperature at the center of the liquid surface is presented in Figure 7. For all ethanol contents, the temperature is reduced by the evaporation. The temperature reduction increases with alcohol content, except for the lowest alcohol case, for which the temperature eventually drops below the two intermediate cases ($\text{abv} = 0.15, 0.4$). This is attributed to the different flow field, as seen in Figure 2, where the cooled vapor travels along the surface to the center of the glass before is rises.

The heat of vaporization of ethanol and water is compared in Figure 8. It can be seen that water has a significantly higher heat of vaporization than ethanol. On a molar basis, the heat of vaporization is similar for the two species, with water having a 4% higher value at 25°C. On a mass basis, as in Figure 8, it is evident that water, despite being a small molecule, has an unusually high heat of vaporization. This can be attributed to the hydrogen bonds of water.

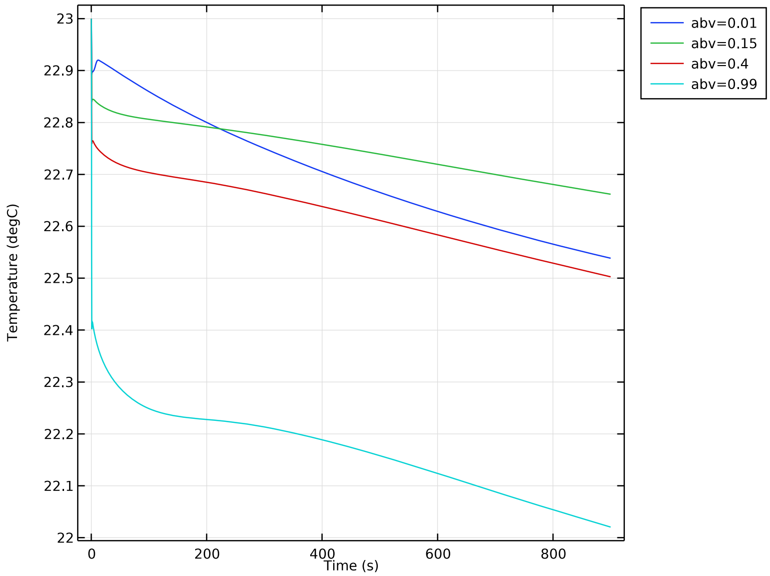


Figure 7: Temperature at the surface center for each abv.

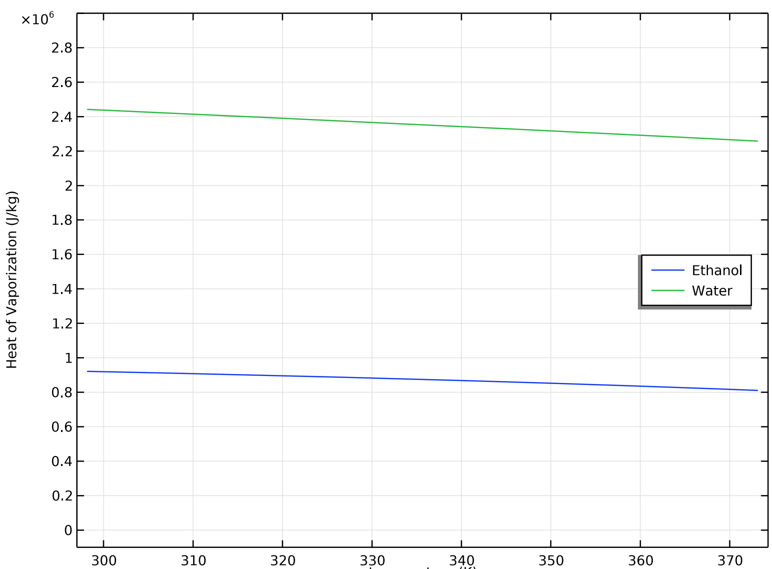


Figure 8: The heat of vaporization for ethanol and water varying with temperature.

References

1. D. Wollan, D.T. Pham, and K.L.Wilkinson, “Changes in Wine Ethanol Content Due to Evaporation from Wine Glasses and Implications for Sensory Analysis,” *J. Agric. Food Chem.*, vol. 64, no. 40, pp. 7569–7575, 2016.

Application Library path: `Chemical_Reaction_Engineering_Module/Thermodynamics/ethanol_water_evaporation`

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 **Chemical Species Transport>Vapor Flow>Laminar Flow, Concentrated Species**.
3. Click **Add**.
4. In the **Number of species** text field, type `3`.
5. In the **Mass fractions (1)** table, enter the following settings:

wEth
wW
wN2

6. In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.
7. Click **Add**.
8. Click **Study**.
9. In the **Select Study** tree, select **General Studies>Time Dependent**.
10. Click **Done**.

Import model parameters from a file.

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `ethanol_water_evaporation_parameters.txt`.

In this model, a **Vapor-Liquid System** that contains compounds of water, ethanol, and nitrogen will be added from **Thermodynamics**. Thermodynamics functions from both vapor and liquid phases are created automatically based on requirements of mass transport and the saturated vapor from the vapor-liquid equilibrium on the boundary.

In the **Physics** toolbar, click **Thermodynamics** and choose **Thermodynamic System**.

SELECT SYSTEM

1 Go to the **Select System** window.

2 From the **Phase** list, choose **Vapor-liquid**.

3 Click **Next** in the window toolbar.

SELECT SPECIES

1 Go to the **Select Species** window.

2 In the **Species** list, select **ethanol (64-17-5, C2H6O)**.

3 Click **Add Selected**.

4 In the **Species** list, select **nitrogen (7727-37-9, N2)**.

5 Click **Add Selected**.

6 In the **Species** list, select **water (7732-18-5, H2O)**.

7 Click **Add Selected**.

8 Click **Next** in the window toolbar.

Select the **UNIQUAC** for the **Liquid phase model**, keep the default model **Soave-Redlich Kwong** for the **Gas phase model**.

SELECT THERMODYNAMIC MODEL

1 Go to the **Select Thermodynamic Model** window.

2 From the list, choose **UNIQUAC**.

3 Click **Finish** in the window toolbar.

GLOBAL DEFINITIONS

Vapor-Liquid System 1 (pp1)

A **Chemistry** node can be generated from **Vapor-Liquid System 1**.

1 In the **Model Builder** window, under **Global Definitions>Thermodynamics** right-click **Vapor-Liquid System 1 (pp1)** and choose **Generate Chemistry**.

SELECT SPECIES

1 Go to the **Select Species** window.

2 Click **Add All**.

3 Click **Next** in the window toolbar.

CHEMISTRY SETTINGS

1 Go to the **Chemistry Settings** window.

2 From the **Mass transfer** list, choose **Concentrated species**.

3 Click **Finish** in the window toolbar.

CHEMISTRY (CHEM)

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

2 In the **Settings** window for **Chemistry**, locate the **Species Matching** section.

3 From the **Species solved for** list, choose **Transport of Concentrated Species in Vapor**.

4 Find the **Bulk species** subsection. In the table, enter the following settings:

Species	Type	Mass fraction	Value (1)	From Thermodynamics
C2H6O	Free species	wEth	Solved for	C2H6O
H2O	Free species	wW	Solved for	H2O
N2	Free species	wN2	Solved for	N2

GEOMETRY 1

Now, define the **Geometry** by inserting it from a file.

1 In the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `ethanol_water_evaporation_geom_sequence.mph`.

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

MULTIPHYSICS

The **Chemistry** interface can be coupled to **Heat Transfer in Fluids** by the **Reacting Flow** Multiphysics Coupling.

Reacting Flow 1 (nirf1)

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Multiphysics** node, then click **Reacting Flow 1 (nirf1)**.
2. In the **Settings** window for **Reacting Flow**, locate the **Coupled Interfaces** section.
3. From the **Chemistry (optional)** list, choose **Chemistry (chem)**.
4. From the **Heat transfer (optional, requires Chemistry)** list, choose **Heat Transfer in Fluids (ht)**.

Add an average function on the boundary where a **Vapor-Liquid Interface** feature will be added later.

DEFINITIONS

Average 1 (aveop1)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.
2. In the **Settings** window for **Average**, locate the **Source Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 8 only.

Import variable definitions from a file.

Vapor-liquid interface variables

1. In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
2. In the **Settings** window for **Variables**, type `Vapor-liquid interface variables` in the **Label** text field.
3. Locate the **Variables** section. Click the **Load** button. From the menu, choose **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `ethanol_water_evaporation_interface_variables.txt`.

   At this stage, the variable expressions are colored yellow. The variable definitions will appear normal when the **Vapor-Liquid Interface** feature is set later. The average function`aveop1` turns the local variable (at the boundary) to a global expression. Then, the expression is defined as a variable on the entire model.

GLOBAL DEFINITIONS

Vapor-Liquid System 1 (pp1)

Now it is time to generate the materials. First, generate the materials that the beverage consists of. These could be generated from **Vapor-Liquid System 1** under **Thermodynamics**.

1. In the **Model Builder** window, under **Global Definitions>Thermodynamics** right-click **Vapor-Liquid System 1 (pp1)** and choose **Generate Material**.

SELECT PHASE

1. Go to the **Select Phase** window.
2. From the list, choose **Liquid**.
3. Click **Next** in the window toolbar.

The beverage contains two species only: water and ethanol.

SELECT SPECIES

1. Go to the **Select Species** window.
2. Click **Remove All**.
3. In the list, choose **ethanol** and **water**.
4. Click **Add Selected**.
5. Find the **Material composition** subsection. Click the **Mass fraction** button.
6. Click **Next** in the window toolbar.

SELECT PROPERTIES

1. Go to the **Select Properties** window.
2. Click **Next** in the window toolbar.

DEFINE MATERIAL

1. Go to the **Define Material** window.
2. Click **Finish** in the window toolbar.

MATERIALS

Liquid: ethanol-water 1 (pp1mat1)

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Materials** node, then click **Liquid: ethanol-water 1 (pp1mat1)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

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

4 Locate the **Material Contents** section. Find the **Local properties** subsection. In the table, enter the following settings:

Name	Expression	Unit	Description	Property group
xw1	`wEthLiq`		Mass fraction, ethanol	Basic
xw2	`wWLiq`		Mass fraction, water	Basic

The glass is made of silica glass, while the table is made of wood. These materials are found in the **Material Library**.

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 **Search** text field, type `silica glass`.

4 Click **Search**.

5 In the tree, select **Built-in>Silica glass**.

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

7 In the **Search** text field, type `wood (pine)`.

8 Click **Search**.

9 In the tree, select **Building>Wood (pine)**.

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

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

MATERIALS

Silica glass (mat1)

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

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

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

Wood (pine) (mat2)

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

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

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

TRANSPORT OF CONCENTRATED SPECIES IN VAPOR (TCS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Concentrated Species in Vapor (tcs)**.

2 In the **Settings** window for **Transport of Concentrated Species in Vapor**, locate the **Domain Selection** section.

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

4 Locate the **Species** section. From the **From mass constraint** list, choose **wN2**.

Transport Properties 1

1 In the **Model Builder** window, expand the **Transport of Concentrated Species in Vapor (tcs)** node, then click **Transport Properties 1**.

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

3 From the ρ list, choose **Density (chem)**.

4 Locate the **Diffusion** section. In the table, enter the following settings:

Species 1	Species 2	Diffusivity	Diffusion coefficient (m^2/s)
wEth	wW	Maxwell-Stefan diffusivity , C2H6O-H2O (chem)	comp1.chem.D_C2H6O_H2O
wEth	wN2	Maxwell-Stefan diffusivity , C2H6O-N2 (chem)	comp1.chem.D_C2H6O_N2
wW	wN2	Maxwell-Stefan diffusivity , H2O-N2 (chem)	comp1.chem.D_H2O_N2

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 $\omega_{0,\mathrm{wEth}}$ text field, type `wEth0`.

4 In the $\omega_{0,\mathrm{wW}}$ text field, type `wW0`.

Remove the default feature **Vapor Inflow** because there is no vapor inflow in this model.

Vapor Inflow 1

In the **Model Builder** window, right-click **Vapor Inflow 1** and choose **Delete**.

Vapor–Liquid Interface 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Transport of Concentrated Species in Vapor (tcs)** click **Vapor–Liquid Interface 1**.

2 Select Boundary 8 only.

3. In the **Settings** window for **Vapor–Liquid Interface**, locate the **Vapor Equilibrium** section.
4. From the **Liquid** list, choose **Thermochemistry coupling**.
5. From the **Chemistry** list, choose **Chemistry (chem)**.
6. Find the **Evaporating/condensing species** subsection. Select the **wEth, C2H6O (ethanol)** check box.
7. Select the **wW, H2O (water)** check box.
8. Locate the **Liquid Phase** section. From the **Liquid phase concentration type** list, choose **Volume fraction**.
9. In the table, enter the following settings:

Species	Volume fraction (1)
C2H6O (ethanol)	abv
N2 (nitrogen)	0
H2O (water)	1-abv

10. Click the **Show More Options** button in the **Model Builder** toolbar.
11. In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.
12. Click **OK**.
13. In the **Settings** window for **Vapor–Liquid Interface**, click to expand the **Constraint Settings** section.
14. From the **Constraint** list, choose **Weak constraints**.

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 **Vapor/Air**.
4. Locate the **Physical Model** section. From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

   The compressible flow for Ma less than 0.3 indicates that the inlet and outlet conditions may not be suitable for transonic or supersonic flow.
5. Select the **Include gravity** check box.
6. In the p_{ref} text field, type `p0`.

A **Vapor-Liquid Slip** at the surface interface should be accounted for.

Vapor-Liquid Slip

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

2 In the **Settings** window for **Wall**, type `Vapor-Liquid Slip` in the **Label** text field.

3 Select Boundary 8 only.

4 Locate the **Boundary Condition** section. From the **Wall condition** list, choose **Navier slip**.

HEAT TRANSFER IN FLUIDS (HT)

Initial Values 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)> Heat Transfer in Fluids (ht)** node, then click **Initial Values 1**.

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

3 In the T text field, type `T0`.

Solid 1

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

   Choosing a **Solid** domain creates a difference in heat transfer between beverage, glass, table, and air.

2 Select Domains 1–3 only.

To demonstrate the evaporation heat from the surface of the beverage, a **Boundary Heat Source** is used.

Boundary Heat Source 1

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

2 In the **Settings** window for **Boundary Heat Source**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Vapor-Liquid Surface**.

4 Locate the **Boundary Heat Source** section. From the Q_b list, choose **Evaporative heat flux (tcs/vl1)**.

Temperature 1

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

2 Select Boundary 2 only.

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

4 In the T_0 text field, type `T0`.

Now, create the **Mesh**.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Size 3

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

2 Drag and drop **Size 3** below **Size 2**.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

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

5 Select Boundary 32 only.

6 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

7 From the **Predefined** list, choose **Finer**.

8 Click the **Custom** button.

9 Locate the **Element Size Parameters** section.

10 Select the **Maximum element size** check box. In the associated text field, type `0.01`.

Size 4

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

2 Drag and drop **Size 4** below **Size 3**.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

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

5 Select Boundaries 35 and 36 only.

6 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

7 From the **Predefined** list, choose **Finer**.

8 Click the **Custom** button.

9 Locate the **Element Size Parameters** section.

10 Select the **Maximum element size** check box. In the associated text field, type `0.75[cm]`.

Size 1

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.

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

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

4 Select Domain 5 only.

5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

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

7 Click the **Custom** button.

8 Locate the **Element Size Parameters** section.

9 Select the **Maximum element growth rate** check box. In the associated text field, type `1.15`.

Boundary Layers 1

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

2 Select Domains 3–5 only.

Boundary Layer Properties 1

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

2 Select Boundaries 7–22, 24–27, and 29–34 only.

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

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

5 In the **Thickness adjustment factor** text field, type `2`.

6 Click **Build Selected**.

Size 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Size 1** and choose **Delete**.

Size 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size 2**.

2 Select Boundaries 7, 9–23, 25, 26, 28, 31, 32, and 34 only.

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

4 From the **Calibrate for** list, choose **Fluid dynamics**.

5 From the **Predefined** list, choose **Extra fine**.

Size 5

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

2 Drag and drop **Size 5** below **Size 4**.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

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

5 Select Boundary 7 only.

6 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

7 From the **Predefined** list, choose **Extra fine**.

Free Triangular 2

1 In the **Mesh** toolbar, click **Free Triangular**.

2 Drag and drop below **Size**.

3 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

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

5 Select Domain 4 only.

Size 1

1 Right-click **Free Triangular 2** and choose **Size**.

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

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 From the **Predefined** list, choose **Extra fine**.

5 Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `0.0015`.

Size 2

1 In the **Model Builder** window, right-click **Free Triangular 2** and choose **Size**.

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

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

4 Select Point 19 only.

5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

6 From the **Predefined** list, choose **Extra fine**.

7 Click the **Custom** button.

8 Locate the **Element Size Parameters** section.

9 Select the **Maximum element size** check box. In the associated text field, type `3e-4`.

10 Click **Build Selected**.

Size 3

1 Right-click **Free Triangular 2** and choose **Size**.

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

3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundaries 25 and 26 only.
5 Locate the **Element Size** section. Click the **Custom** button.
6 Locate the **Element Size Parameters** section.
7 Select the **Curvature factor** check box. In the associated text field, type `0.1`.
8 Click **Build Selected**.
9 Click **Build All**.

By using a **Parametric Sweep**, the results can be observed for several abv values.

STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.
2 In the **Model Builder** window, expand the **Study 1** node, then click **Parametric Sweep**.
3 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
4 Click **Add**.
5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
abv (0.35 Alcohol by volume)	0.01 0.15 0.4 0.99	

Solution 1 (sol1)

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

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.
2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3 In the **Output times** text field, type `range(0,0.1,1) range(1.25,0.25,4) range(5,1,20) range(25,5,120) range(130,10,240) range(270,30,900)`.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Temperature (comp1.T)**.
2 In the **Settings** window for **Field**, locate the **Scaling** section.
3 From the **Method** list, choose **Manual**.

4 In the **Scale** text field, type `T0`.
5 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Velocity field (comp1.u)**.
6 In the **Settings** window for **Field**, locate the **Scaling** section.
7 From the **Method** list, choose **Manual**.
8 In the **Scale** text field, type `0.01`.
9 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Mass fraction (comp1.wEth)**.
10 In the **Settings** window for **Field**, locate the **Scaling** section.
11 From the **Method** list, choose **Manual**.
12 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Mass fraction (comp1.wW)**.
13 In the **Settings** window for **Field**, locate the **Scaling** section.
14 From the **Method** list, choose **Manual**.

Use an initial time step of 1 ms. This is small compared to the time scale and will make it easy for the transient solver to start.

1 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Time-Dependent Solver 1**.
2 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
3 Select the **Initial step** check box.
4 In the **Study** toolbar, click **Get Initial Value** to generate the default datasets and plots. This means that a plot to be shown while solving can be chosen as well. For that purpose, create a **Plot Group** showing both velocity, temperature, and mass fractions.

RESULTS

2D Plot Group 11

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

Mirror 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.
2 In the **Settings** window for **Mirror 2D**, click to expand the **Advanced** section.
3 Select the **Remove elements on the symmetry axis** check box.

Array: Velocity, Temperature, and Mass Fractions

1 In the **Model Builder** window, under **Results** click **2D Plot Group 11**.

2 In the **Settings** window for **2D Plot Group**, type `Array: Velocity, Temperature, and Mass Fractions` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.

4 Click to expand the **Title** section. Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

5 Select the **Show units** check box.

6 Click to expand the **Number Format** section. Select the **Manual color legend settings** check box.

7 From the **Notation** list, choose **Scientific**.

8 Click to expand the **Plot Array** section. Select the **Enable** check box.

9 From the **Array shape** list, choose **Square**.

10 From the **Order** list, choose **Column-major**.

11 In the **Relative row padding** text field, type `0.1`.

12 In the **Relative column padding** text field, type `0.1`.

Velocity

1 Right-click **Array: Velocity, Temperature, and Mass Fractions** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Velocity` in the **Label** text field.

3 Click to expand the **Expression** section. In the **Expression** text field, type `spf.U`.

Mass Fraction, Water

1 Right-click **Array: Velocity, Temperature, and Mass Fractions** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Mass Fraction, Water` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `wW`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Wave>Disco** in the tree.

6 Click **OK**.

7 In the **Settings** window for **Surface**, click to expand the **Plot Array** section.

8 Select the **Manual indexing** check box.

9 In the **Row index** text field, type `-1`.

Temperature

1 Right-click **Array: Velocity, Temperature, and Mass Fractions** and choose **Surface**.

2. In the **Settings** window for **Surface**, type `Temperature` in the **Label** text field.
3. Locate the **Expression** section. In the **Expression** text field, type `T`.
4. From the **Unit** list, choose **degC**.
5. Locate the **Coloring and Style** section. Click **Change Color Table**.
6. In the **Color Table** dialog box, select **Thermal>HeatCamera** in the tree.
7. Click **OK**.
8. In the **Settings** window for **Surface**, locate the **Plot Array** section.
9. Select the **Manual indexing** check box.
10. In the **Column index** text field, type `1`.

Mass Fraction, Water

In the **Model Builder** window, right-click **Mass Fraction, Water** and choose **Duplicate**.

Mass Fraction, Ethanol

1. In the **Model Builder** window, under **Results>Array: Velocity, Temperature, and Mass Fractions** click **Mass Fraction, Water 1**.
2. In the **Settings** window for **Surface**, type `Mass Fraction, Ethanol` in the **Label** text field.
3. Locate the **Expression** section. In the **Expression** text field, type `wEth`.
4. Locate the **Plot Array** section. Select the **Manual indexing** check box.
5. In the **Column index** text field, type `1`.
6. In the **Array: Velocity, Temperature, and Mass Fractions** toolbar, click **Plot**.

Array: Velocity, Temperature, and Mass Fractions

1. In the **Model Builder** window, click **Array: Velocity, Temperature, and Mass Fractions**.
2. In the **Settings** window for **2D Plot Group**, locate the **Color Legend** section.
3. From the **Position** list, choose **Right double**.

Arrow Surface 1

1. In the **Model Builder** window, right-click **Array: Velocity, Temperature, and Mass Fractions** and choose **Arrow Surface**.
2. In the **Settings** window for **Arrow Surface**, locate the **Arrow Positioning** section.
3. Find the **x grid points** subsection. In the **Points** text field, type `12`.
4. Find the **y grid points** subsection. In the **Points** text field, type `12`.
5. Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.
6. From the **Arrow length** list, choose **Logarithmic**.

7 In the **Range quotient** text field, type `20`.

8 From the **Color** list, choose **White**.

9 Click to expand the **Plot Array** section. Select the **Manual indexing** check box.

10 In the **Array: Velocity, Temperature, and Mass Fractions** toolbar, click **Plot**.

Annotation: Velocity

1 In the **Model Builder** window, right-click **Array: Velocity, Temperature, and Mass Fractions** and choose **Annotation**.

2 In the **Settings** window for **Annotation**, type `Annotation: Velocity` in the **Label** text field.

3 Locate the **Annotation** section. In the **Text** text field, type `Velocity`.

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

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

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

7 Click to expand the **Plot Array** section. Clear the **Belongs to array** check box.

8 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Lower left**.

9 Right-click **Annotation: Velocity** and choose **Duplicate**.

Annotation: Temperature

1 In the **Model Builder** window, under **Results>Array: Velocity, Temperature, and Mass Fractions** click **Annotation: Velocity 1**.

2 In the **Settings** window for **Annotation**, type `Annotation: Temperature` in the **Label** text field.

3 Locate the **Annotation** section. In the **Text** text field, type `Temperature`.

4 Locate the **Position** section. In the **x** text field, type `0.6`.

5 Right-click **Annotation: Temperature** and choose **Duplicate**.

Annotation: Water

1 In the **Model Builder** window, under **Results>Array: Velocity, Temperature, and Mass Fractions** click **Annotation: Temperature 1**.

2 In the **Settings** window for **Annotation**, type `Annotation: Water` in the **Label** text field.

3 Locate the **Annotation** section. In the **Text** text field, type `Water`.

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

5 In the **y** text field, type `-0.7`.

6 Right-click **Annotation: Water** and choose **Duplicate**.

Annotation: Ethanol

1 In the **Model Builder** window, under **Results>Array: Velocity, Temperature, and Mass Fractions** click **Annotation: Water 1**.

2 In the **Settings** window for **Annotation**, type `Annotation: Ethanol` in the **Label** text field.

3 Locate the **Annotation** section. In the **Text** text field, type `Ethanol`.

4 Locate the **Position** section. In the **x** text field, type `0.6`.

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

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**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Array: Velocity, Temperature, and Mass Fractions**.

5 From the **Update at** list, choose **Time steps taken by solver**.

Parametric Sweep

1 In the **Model Builder** window, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Output While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Array: Velocity, Temperature, and Mass Fractions**.

Now compute the solutions for the four different alcohol contents.

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

Point the mirror dataset to the parametric solutions to be able to plot results from all compositions.

RESULTS

Mirror 2D 1

1 In the **Model Builder** window, under **Results>Datasets** click **Mirror 2D 1**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

Now create a plot that shows the velocity in the vapor phase for all four compositions at the same time step.

Array: Velocity all cases

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
2 In the **Settings** window for **2D Plot Group**, type `Array: Velocity all cases` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.
4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
5 In the **Parameter indicator** text field, type `Time =eval(t) s`.
6 Click to expand the **Plot Array** section. Select the **Enable** check box.
7 From the **Array shape** list, choose **Square**.
8 In the **Relative row padding** text field, type `0.1`.
9 In the **Relative column padding** text field, type `0.1`.

abv = 0.01

1 Right-click **Array: Velocity all cases** and choose **Surface**.
2 In the **Settings** window for **Surface**, type `abv = 0.01` in the **Label** text field.
3 Locate the **Expression** section. In the **Expression** text field, type `withsol('sol2',spf.U,setval(abv,0.01,t,t))`.
4 Click to expand the **Plot Array** section. In the **Array: Velocity all cases** toolbar, click **Plot**.
5 Right-click **abv = 0.01** and choose **Duplicate**.

abv = 0.15

1 In the **Model Builder** window, under **Results>Array: Velocity all cases** click **abv = 0.01.1**.
2 In the **Settings** window for **Surface**, type `abv = 0.15` in the **Label** text field.
3 Locate the **Expression** section. In the **Expression** text field, type `withsol('sol2',spf.U,setval(abv,0.15,t,t))`.
4 Locate the **Plot Array** section. Select the **Manual indexing** check box.
5 In the **Row index** text field, type `-1`.
6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **abv = 0.01**.
7 In the **Array: Velocity all cases** toolbar, click **Plot**.
8 Right-click **abv = 0.15** and choose **Duplicate**.

abv = 0.4

1 In the **Model Builder** window, under **Results>Array: Velocity all cases** click **abv = 0.15.1**.
2 In the **Settings** window for **Surface**, type `abv = 0.4` in the **Label** text field.
3 Locate the **Expression** section. In the **Expression** text field, type `withsol('sol2', spf.U,setval(abv,0.4,t,t))`.
4 Locate the **Plot Array** section. In the **Row index** text field, type `0`.
5 In the **Column index** text field, type `1`.
6 In the **Array: Velocity all cases** toolbar, click **Plot**.
7 Right-click **abv = 0.4** and choose **Duplicate**.

abv = 0.99

1 In the **Model Builder** window, under **Results>Array: Velocity all cases** click **abv = 0.4.1**.
2 In the **Settings** window for **Surface**, type `abv = 0.99` in the **Label** text field.
3 Locate the **Expression** section. In the **Expression** text field, type `withsol('sol2', spf.U,setval(abv,0.99,t,t))`.
4 Locate the **Plot Array** section. In the **Row index** text field, type `-1`.
5 In the **Array: Velocity all cases** toolbar, click **Plot**.

Arrow Surface: abv = 0.01

1 In the **Model Builder** window, right-click **Array: Velocity all cases** and choose **Arrow Surface**.
2 In the **Settings** window for **Arrow Surface**, type `Arrow Surface: abv = 0.01` in the **Label** text field.
3 Locate the **Expression** section. In the **x-component** text field, type `withsol('sol2',u, setval(abv,0.01,t,t))`.
4 In the **y-component** text field, type `withsol('sol2',w,setval(abv,0.01,t,t))`.
5 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `12`.
6 Find the **y grid points** subsection. In the **Points** text field, type `12`.
7 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.
8 From the **Arrow length** list, choose **Logarithmic**.
9 In the **Range quotient** text field, type `20`.
10 From the **Color** list, choose **White**.
11 Locate the **Plot Array** section. Select the **Manual indexing** check box.

12 Right-click **Arrow Surface: abv = 0.01** and choose **Duplicate**.

Arrow Surface: abv = 0.15

1 In the **Model Builder** window, under **Results>Array: Velocity all cases** click **Arrow Surface: abv = 0.01.1**.

2 In the **Settings** window for **Arrow Surface**, type `Arrow Surface: abv = 0.15` in the **Label** text field.

3 Locate the **Expression** section. In the **x-component** text field, type `withsol('sol2',u, setval(abv,0.15,t,t))`.

4 In the **y-component** text field, type `withsol('sol2',w,setval(abv,0.15,t,t))`.

5 Locate the **Plot Array** section. In the **Row index** text field, type `-1`.

6 Right-click **Arrow Surface: abv = 0.15** and choose **Duplicate**.

Arrow Surface: abv = 0.4

1 In the **Model Builder** window, under **Results>Array: Velocity all cases** click **Arrow Surface: abv = 0.15.1**.

2 In the **Settings** window for **Arrow Surface**, type `Arrow Surface: abv = 0.4` in the **Label** text field.

3 Locate the **Expression** section. In the **x-component** text field, type `withsol('sol2',u, setval(abv,0.4,t,t))`.

4 In the **y-component** text field, type `withsol('sol2',w,setval(abv,0.4,t,t))`.

5 Locate the **Plot Array** section. In the **Row index** text field, type `0`.

6 In the **Column index** text field, type `1`.

7 Right-click **Arrow Surface: abv = 0.4** and choose **Duplicate**.

Arrow Surface: abv = 0.99

1 In the **Model Builder** window, under **Results>Array: Velocity all cases** click **Arrow Surface: abv = 0.4.1**.

2 In the **Settings** window for **Arrow Surface**, type `Arrow Surface: abv = 0.99` in the **Label** text field.

3 Locate the **Expression** section. In the **x-component** text field, type `withsol('sol2',u, setval(abv,0.99,t,t))`.

4 In the **y-component** text field, type `withsol('sol2',w,setval(abv,0.99,t,t))`.

5 Locate the **Plot Array** section. In the **Row index** text field, type `-1`.

6 In the **Array: Velocity all cases** toolbar, click **Plot**.

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

abv = 0.15

1 In the **Model Builder** window, click **abv = 0.15**.
2 In the **Settings** window for **Surface**, click to expand the **Inherit Style** section.
3 From the **Plot** list, choose **abv = 0.01**.
4 In the **Array: Velocity all cases** toolbar, click **Plot**.
5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Annotation: abv = 0.01

1 In the **Model Builder** window, right-click **Array: Velocity all cases** and choose **Annotation**.
2 In the **Settings** window for **Annotation**, type `Annotation: abv = 0.01` in the **Label** text field.
3 Locate the **Annotation** section. In the **Text** text field, type `abv = 0.01`.
4 Locate the **Position** section. In the **x** text field, type `-0.5`.
5 In the **y** text field, type `0.5`.
6 Locate the **Coloring and Style** section. Clear the **Show point** check box.
7 From the **Anchor point** list, choose **Lower left**.
8 Locate the **Plot Array** section. Clear the **Belongs to array** check box.
9 In the **Array: Velocity all cases** toolbar, click **Plot**.
10 Right-click **Annotation: abv = 0.01** and choose **Duplicate**.

Annotation: abv = 0.4

1 In the **Model Builder** window, under **Results>Array: Velocity all cases** click **Annotation: abv = 0.01.1**.
2 In the **Settings** window for **Annotation**, type `Annotation: abv = 0.4` in the **Label** text field.
3 Locate the **Annotation** section. In the **Text** text field, type `abv = 0.4`.
4 Locate the **Position** section. In the **x** text field, type `0.6`.
5 In the **Array: Velocity all cases** toolbar, click **Plot**.
6 Right-click **Annotation: abv = 0.4** and choose **Duplicate**.

Annotation: abv = 0.15

1 In the **Model Builder** window, under **Results>Array: Velocity all cases** click **Annotation: abv = 0.4.1**.
2 In the **Settings** window for **Annotation**, type `Annotation: abv = 0.15` in the **Label** text field.

3 Locate the **Annotation** section. In the **Text** text field, type `abv = 0.15`.

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

5 In the **y** text field, type `-0.7`.

6 In the **Array: Velocity all cases** toolbar, click **Plot**.

7 Right-click **Annotation: abv = 0.15** and choose **Duplicate**.

Annotation: abv = 0.99

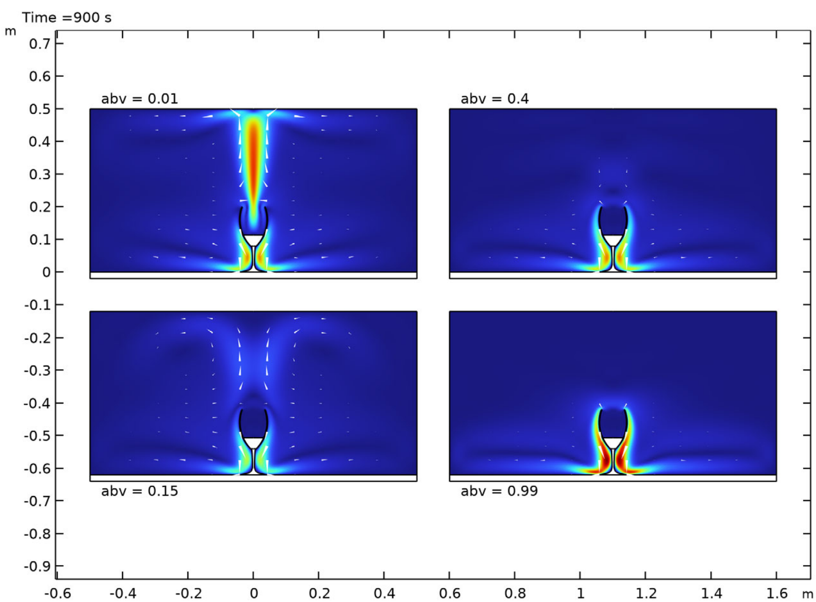
1 In the **Model Builder** window, under **Results>Array: Velocity all cases** click **Annotation: abv = 0.15.1**.

2 In the **Settings** window for **Annotation**, type `Annotation: abv = 0.99` in the **Label** text field.

3 Locate the **Annotation** section. In the **Text** text field, type `abv = 0.99`.

4 Locate the **Position** section. In the **x** text field, type `0.6`.

5 In the **Array: Velocity all cases** toolbar, click **Plot**.



Now, create line graphs evaluating the mass flux due to evaporation. To obtain these, integrate the normal total fluxes of ethanol and water along the vapor-liquid surface.

Evaluation: Water Flux

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

2 In the **Settings** window for **Evaluation Group**, type `Evaluation: Water Flux` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

Line Integration 1

1 Right-click **Evaluation: Water Flux** and choose **Integration>Line Integration**.

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

3 From the **Table columns** list, choose **abv**.

4 Locate the **Selection** section. From the **Selection** list, choose **Vapor-Liquid Surface**.

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

Expression	Unit	Description
-tcs.ntflux_wW	g/h	

6 In the **Evaluation: Water Flux** toolbar, click **Evaluate**.

7 Right-click **Line Integration 1** and choose **Duplicate**.

Line Integration 2

1 In the **Model Builder** window, click **Line Integration 2**.

2 In the **Settings** window for **Line Integration**, locate the **Data Series Operation** section.

3 From the **Transformation** list, choose **Integral**.

4 Select the **Cumulative** check box.

5 In the **Evaluation: Water Flux** toolbar, click **Evaluate**.

1D Plot Group 13

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

Table Graph 1

1 Right-click **1D Plot Group 13** and choose **Table Graph**.

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

3 From the **Source** list, choose **Evaluation group**.

4 From the **Evaluation group** list, choose **Evaluation: Water Flux**.

5 From the **Plot columns** list, choose **Manual**.

6 In the **Columns** list, choose **abv=0.01, -tcs.ntflux_wW (g/h)**, **abv=0.15, -tcs.ntflux_wW (g/ h)**, **abv=0.4, -tcs.ntflux_wW (g/h)**, and **abv=0.99, -tcs.ntflux_wW (g/h)**.

7 In the **1D Plot Group 13** toolbar, click **Plot**.

8 Right-click **Table Graph 1** and choose **Duplicate**.

Water Mass Flux

1 In the **Model Builder** window, under **Results** click **1D Plot Group 13**.

2 In the **Settings** window for **1D Plot Group**, type `Water Mass Flux` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Vapor Mass Flux (g/h)`.

6 Select the **Two y-axes** check box.

7 Select the **Secondary y-axis label** check box. In the associated text field, type `Accumulated Mass (g)`.

8 In the table, select the **Plot on secondary y-axis** check box for **Table Graph 2**.

9 Locate the **Legend** section. From the **Layout** list, choose **Outside graph axis area**.

10 From the **Position** list, choose **Top**.

11 In the **Number of rows** text field, type `2`.

Table Graph 2

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

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

3 In the **Columns** list, choose **abv=0.01, Cumulative integral: -tcs.ntflux_wW (g)**, **abv=0.15, Cumulative integral: -tcs.ntflux_wW (g)**, **abv=0.4, Cumulative integral: -tcs.ntflux_wW (g)**, and **abv=0.99, Cumulative integral: -tcs.ntflux_wW (g)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 From the **Color** list, choose **Cycle (reset)**.

6 In the **Water Mass Flux** toolbar, click **Plot**.

Water Mass Flux

1 In the **Model Builder** window, click **Water Mass Flux**.

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

3 Select the **Manual axis limits** check box.

4 In the **y minimum** text field, type `-0.1`.

5 In the **y maximum** text field, type `0.2`.

6 In the **Water Mass Flux** toolbar, click **Plot**.

Evaluation: Water Flux

In the **Model Builder** window, right-click **Evaluation: Water Flux** and choose **Duplicate**.

Evaluation: Ethanol Flux

1 In the **Model Builder** window, expand the **Results>Evaluation: Water Flux 1** node, then click **Evaluation: Water Flux 1**.

2 In the **Settings** window for **Evaluation Group**, type `Evaluation: Ethanol Flux` in the **Label** text field.

Line Integration 1

1 In the **Model Builder** window, click **Line Integration 1**.

2 In the **Settings** window for **Line Integration**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Transport of Concentrated Species in Vapor>Species wEth>Fluxes>tcs.ntflux_wEth - Normal total flux - kg/(m²·s)**.

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

Expression	Unit	Description
`-tcs.ntflux_wEth`	`g/h`	

Line Integration 2

1 In the **Model Builder** window, click **Line Integration 2**.

2 In the **Settings** window for **Line Integration**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **tcs.ntflux_wEth - Normal total flux - kg/(m²·s)**.

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

Expression	Unit	Description
`-tcs.ntflux_wEth`	`g/h`	

4 In the **Evaluation: Ethanol Flux** toolbar, click = **Evaluate**.

Water Mass Flux

In the **Model Builder** window, under **Results** right-click **Water Mass Flux** and choose **Duplicate**.

Ethanol Mass Flux

1 In the **Model Builder** window, under **Results** click **Water Mass Flux 1**.

2 In the **Settings** window for **1D Plot Group**, type `Ethanol Mass Flux` in the **Label** text field.

Table Graph 1

1 In the **Model Builder** window, expand the **Ethanol Mass Flux** node, then click **Table Graph 1**.

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

3 From the **Evaluation group** list, choose **Evaluation: Ethanol Flux**.

Table Graph 2

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

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

3 From the **Evaluation group** list, choose **Evaluation: Ethanol Flux**.

4 In the **Ethanol Mass Flux** toolbar, click **Plot**.

Ethanol Mass Flux

1 In the **Model Builder** window, click **Ethanol Mass Flux**.

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

3 Select the **Manual axis limits** check box.

4 In the **y minimum** text field, type `0`.

5 In the **y maximum** text field, type `1`.

6 In the **Secondary y minimum** text field, type `0`.

7 In the **Secondary y maximum** text field, type `0.085`.

Evaluation: Kinetic Energy

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

2 In the **Settings** window for **Evaluation Group**, type `Evaluation: Kinetic Energy` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

Surface Integration 1

1 Right-click **Evaluation: Kinetic Energy** and choose **Integration>Surface Integration**.

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

3 From the **Table columns** list, choose **abv**.

4 Locate the **Selection** section. From the **Selection** list, choose **Vapor/Air**.

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

Expression	Unit	Description
spf.rho*spf.U	N*s	

6 In the **Evaluation: Kinetic Energy** toolbar, click **Evaluate**.

Kinetic Energy

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Kinetic Energy (N*s)`.

6 Locate the **Legend** section. From the **Layout** list, choose **Outside graph axis area**.

Table Graph 1

1 Right-click **Kinetic Energy** and choose **Table Graph**.

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

3 From the **Source** list, choose **Evaluation group**.

4 From the **Evaluation group** list, choose **Evaluation: Kinetic Energy**.

5 From the **Plot columns** list, choose **All excluding x-axis**.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 In the **Kinetic Energy** toolbar, click **Plot**.

8 Click **Plot**.

Surface 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 `Surface Temperature` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 Locate the **Legend** section. From the **Layout** list, choose **Outside graph axis area**.

Point Graph 1

1 In the **Model Builder** window, right-click **Surface Temperature** and choose **Point Graph**.

2 Select Point 4 only.

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

4 In the **Expression** text field, type `T`.

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

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Automatic**.

8 Find the **Include** subsection. Clear the **Point** check box.

9 In the **Surface Temperature** toolbar, click **Plot**.

Surface Temperature

In the **Model Builder** window, right-click **Surface Temperature** and choose **Duplicate**.

Surface Concentration Ethanol

1 In the **Model Builder** window, under **Results** click **Surface Temperature 1**.

2 In the **Settings** window for **1D Plot Group**, type `Surface Concentration Ethanol` in the **Label** text field.

Point Graph 1

1 In the **Model Builder** window, expand the **Surface Concentration Ethanol** node, then click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, locate the **Selection** section.

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

4 Select Point 4 only.

5 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Transport of Concentrated Species in Vapor>Species wEth>tcs.c_wEth - Molar concentration - mol/m³**.

6 In the **Surface Concentration Ethanol** toolbar, click **Plot**.

To show the figure of Glass and Table .

Study 1/Solution 1 (3) (sol1)

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

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

3 From the **Solution** list, choose **Parametric Solutions 1 (sol2)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

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

Revolution Glass

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

2 In the **Settings** window for **Revolution 2D**, type `Revolution Glass` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (3) (sol2)**.

4 Click to expand the **Revolution Layers** section. In the **Start angle** text field, type `-90`.

Glass and Table

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

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

3 From the **Dataset** list, choose **Revolution Glass**.

4 In the **Label** text field, type `Glass and Table`.

5 Locate the **Data** section. From the **Parameter value (abv)** list, choose **0.15**.

6 From the **Time (s)** list, choose **100**.

7 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Volume 1

1 Right-click **Glass and Table** and choose **Volume**.

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

3 From the **Dataset** list, choose **Revolution Glass**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Expression** section. In the **Expression** text field, type `1`.

Material Appearance 1

1 Right-click **Volume 1** and choose **Material Appearance**.

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

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Custom**.

5 Click **Define custom colors**.

6 Set the RGB values to 239, 239, and 235, respectively.

7 Click **Add to custom colors**.

8 Click **Show color palette only** or **OK** on the cross-platform desktop.

9 Click **Define custom colors**.

10 Set the RGB values to 230, 230, and 255, respectively.

11 Click **Add to custom colors**.

12 Click **Show color palette only** or **OK** on the cross-platform desktop.

13 Click **Define custom colors**.

14 Set the RGB values to 230, 230, and 255, respectively.

15 Click **Add to custom colors**.

16 Click **Show color palette only** or **OK** on the cross-platform desktop.

17 Select the **Normal mapping** check box.

18 In the **Normal vector noise scale** text field, type `0.1`.

19 In the **Normal vector noise frequency** text field, type `30`.

20 Select the **Additional color** check box.

21 In the **Noise scale** text field, type `1`.

22 In the **Noise frequency** text field, type `10`.

23 In the **Color blend** text field, type `0.55`.

24 Click **Define custom colors**.

25 Set the RGB values to 16, 31, and 86, respectively.

26 Click **Add to custom colors**.

27 Click **Show color palette only** or **OK** on the cross-platform desktop.

28 In the **Transparency** text field, type `0.5`.

29 In the **Reflectance at normal incidence** text field, type `0.9`.

30 In the **Surface roughness** text field, type `0.25`.

31 In the **Metallic** text field, type `0.85`.

32 In the **Pearl** text field, type `0.05`.

33 In the **Diffuse wrap** text field, type `0.45`.

34 In the **Clear coat** text field, type `0.3`.

35 In the **Reflectance** text field, type `0.75`.

Study 1/Parametric Solutions 1 (3) (sol2)

In the **Model Builder** window, under **Results>Datasets** right-click **Study 1/ Parametric Solutions 1 (3) (sol2)** and choose **Duplicate**.

Selection

1 In the **Model Builder** window, expand the **Results>Datasets>Study 1/ Parametric Solutions 1 (4) (sol2)** node, then click **Selection**.

2 Select Domain 1 only.

Revolution Glass

In the **Model Builder** window, under **Results>Datasets** right-click **Revolution Glass** and choose **Duplicate**.

Revolution Table

1 In the **Model Builder** window, under **Results>Datasets** click **Revolution Glass 1**.

2 In the **Settings** window for **Revolution 2D**, type `Revolution Table` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (4) (sol2)**.

Volume 2

1 In the **Model Builder** window, right-click **Glass and Table** and choose **Volume**.

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

3 From the **Dataset** list, choose **Revolution Table**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Material Appearance 1

1 Right-click **Volume 2** and choose **Material Appearance**.

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

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Wood**.

Cut Plane 1

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

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

3 From the **Dataset** list, choose **Revolution 2D 1**.

Surface 1

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

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

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

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

5 Locate the **Expression** section. In the **Expression** text field, type `spf.U`.

Transparency 1

1 Right-click **Surface 1** and choose **Transparency**.

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

3 In the **Transparency** text field, type `0.4`.

4 In the **Glass and Table** toolbar, click **Plot**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Glass and Table** and choose **Arrow Surface**.

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

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

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

5 Locate the **Expression** section. In the **r-component** text field, type `u`.

6 In the **phi-component** text field, type `v`.

7 In the **z-component** text field, type `w`.

8 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `300`.

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

10 From the **Color** list, choose **Black**.

11 In the **Glass and Table** toolbar, click **Plot**.

Create a plot that evaluates the heat of vaporization of water and ethanol.

GLOBAL DEFINITIONS

In the **Model Builder** window, expand the **Global Definitions>Thermodynamics** node.

Vapor-Liquid System 1 (pp1)

In the **Model Builder** window, expand the **Global Definitions>Thermodynamics>Vapor-Liquid System 1 (pp1)** node.

Heat of vaporization 1 (chempp1dHvap_ethanol, chempp1dHvap_ethanol_Dtemperature)

1 In the **Model Builder** window, expand the **Global Definitions>Thermodynamics>Vapor-Liquid System 1 (pp1)>ethanol** node, then click **Heat of vaporization 1 (chempp1dHvap_ethanol, chempp1dHvap_ethanol_Dtemperature)**.

2 In the **Settings** window for **Species Property**, click **Create Plot**.

GLOBAL DEFINITIONS

Heat of vaporization 2 (chempp1dHvap_water, chempp1dHvap_water_Dtemperature)

1 In the **Model Builder** window, expand the **Global Definitions>Thermodynamics>Vapor-Liquid System 1 (pp1)>water** node, then click **Heat of vaporization 2 (chempp1dHvap_water, chempp1dHvap_water_Dtemperature)**.

2 In the **Settings** window for **Species Property**, click **Create Plot**.

RESULTS

1D Plot Group 19

In the **Model Builder** window, expand the **Results>1D Plot Group 19** node.

Function 1

In the **Model Builder** window, expand the **Results>1D Plot Group 20** node, then click **Function 1**.

Water

1 Drag and drop below **1D Plot Group 19>Function 1**.

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

3 Click to expand the **Legends** section. Select the **Show legends** check box.

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

5 In the table, enter the following settings:

Legends
Water

Ethanol

1 In the **Model Builder** window, under **Results>1D Plot Group 19** click **Function 1**.

2 In the **Settings** window for **Function**, type `Ethanol` in the **Label** text field.

3 Locate the **Legends** section. Select the **Show legends** check box.

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

5 In the table, enter the following settings:

Legends
Ethanol

Heat of Vaporization

1 In the **Model Builder** window, under **Results** click **1D Plot Group 19**.

2 In the **Settings** window for **1D Plot Group**, type `Heat of Vaporization` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Heat of Vaporization (J/kg)`.

6 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

7 Locate the **Axis** section. Select the **Manual axis limits** check box.

8 In the **y minimum** text field, type `-1e5`.

9 In the **y maximum** text field, type `3e6`.

10 In the **Heat of Vaporization** toolbar, click **Plot**.

Appendix — Geometry Modeling Instructions

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

NEW

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

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **2D Axisymmetric**.

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 Click **Clear Table**.

4 In the table, enter the following settings:

Name	Expression	Value	Description
`Hvdom`	`0.5[m]`	0.5 m	`Height of vapor domain`
`Wvdom`	`0.5[m]`	0.5 m	`Width of vapor domain`
`Ttable`	`2[cm]`	0.02 m	`Table thickness`

GEOMETRY 1

Rectangle 1 (r1)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `Wvdom`.

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

Point 1 (pt1)

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

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

3 In the **r** text field, type `0.008`.

4 In the **z** text field, type `0.08`.

Point 2 (pt2)

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

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

3 In the **r** text field, type `0.042`.

4 In the **z** text field, type `0.145`.

Point 1 (pt1), Point 2 (pt2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1**, Ctrl-click to select **Point 1 (pt1)** and **Point 2 (pt2)**.

2 Right-click and choose **Group**.

Point 3 (pt3)

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

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

3 In the **r** text field, type `0.034`.

4 In the **z** text field, type `0.2`.

Point 4 (pt4)

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

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

3 In the **r** text field, type `0.036`.

4 In the **z** text field, type `0.2`.

5 Right-click **Point 4 (pt4)** and choose **Duplicate**.

Point 5 (pt5)

1 In the **Model Builder** window, click **Point 5 (pt5)**.

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

3 In the **r** text field, type `0.04`.

4 In the **z** text field, type `0.145`.

5 Right-click **Point 5 (pt5)** and choose **Duplicate**.

Point 6 (pt6)

1 In the **Model Builder** window, click **Point 6 (pt6)**.

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

3 In the **r** text field, type `0.01`.

4 In the **z** text field, type `0.08`.

5 Right-click **Point 6 (pt6)** and choose **Duplicate**.

Point 7 (pt7)

1 In the **Model Builder** window, click **Point 7 (pt7)**.

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

3 In the **r** text field, type `0.003`.

4 In the **z** text field, type `0.038`.

5 Right-click **Point 7 (pt7)** and choose **Duplicate**.

Point 8 (pt8)

1 In the **Model Builder** window, click **Point 8 (pt8)**.

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

3 In the **r** text field, type `0.042`.

4 In the **z** text field, type `0`.

5 Right-click **Point 8 (pt8)** and choose **Duplicate**.

Point 9 (pt9)

1 In the **Model Builder** window, click **Point 9 (pt9)**.

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

3 In the **r** text field, type `0`.

4 In the **z** text field, type `0.08`.

Group 1: Points

1 In the **Model Builder** window, click **Group 1**.

2 In the **Settings** window for **Group**, type `Group 1: Points` in the **Label** text field.

3 Click **Build All Objects**.

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. In the table, enter the following settings:

r (m)	z (m)
0.01	0.08
0.008	0.079
0.006	0.078
0.005	0.076
0.004	0.064
0.003	0.038
0.003	0.022
0.0048404261469841	0.016
0.0062340328097343425	0.012
0.008117016404867172	0.01
0.01	0.008
0.01396806538105011	0.006
0.042	0
0	0
0	0.08
0.008	0.08

Circular Arc 1 (ca1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.

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

3 From the **Specify** list, choose **Endpoints and radius**.

4 Locate the **Starting Point** section. In the **r** text field, type `0.008`.

5 In the **z** text field, type `0.08`.

6 Locate the **Endpoint** section. In the **r** text field, type `0.034`.

7 In the **z** text field, type `0.2`.

8 Locate the **Radius** section. In the **Radius** text field, type `0.11446`.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **r** text field, type `0.034`.

5 In the **z** text field, type `0.2`.

6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

7 In the **r** text field, type `0.036`.

8 In the **z** text field, type `0.2`.

Circular Arc 1 (ca1)

In the **Model Builder** window, right-click **Circular Arc 1 (ca1)** and choose **Duplicate**.

Circular Arc 2 (ca2)

1 In the **Model Builder** window, click **Circular Arc 2 (ca2)**.

2 In the **Settings** window for **Circular Arc**, locate the **Starting Point** section.

3 In the **r** text field, type `0.01`.

4 Locate the **Endpoint** section. In the **r** text field, type `0.036`.

Union 1 (uni1)

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

2 Select the objects **ca1**, **ca2**, **ls1**, and **pol1** only.

3 In the **Settings** window for **Union**, click **Build All Objects**.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Select the object **uni1** only.

3 In the **Settings** window for **Convert to Solid**, click **Build All Objects**.

Fillet 1 (fil1)

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

2 On the object **csol1**, select Points 16 and 17 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `0.9[mm]`.

Polygon 2 (pol2)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

r (m)	z (m)
0	0.08
0.008	0.08
0.01859369918701851	0.09
0.032	0.1133
0	0.1133
0	0.08

Difference 1 (dif1)

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

2 Select the object **pol2** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **fil1** only.

6 Select the **Keep objects to subtract** check box.

Polygon 3 (pol3)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

r (m)	z (m)
0	0.1133
0.032	0.1133
0.0365	0.124
0	0.124

Difference 1 (dif1)

In the **Model Builder** window, right-click **Difference 1 (dif1)** and choose **Duplicate**.

Difference 2 (dif2)

1 In the **Model Builder** window, click **Difference 2 (dif2)**.

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

3 Click to select the **Activate Selection** toggle button for **Objects to add**.

4 Select the object **pol3** only.

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 `Wvdom`.

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

5 Locate the **Position** section. In the **z** text field, type `-Ttable`.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **r** text field, type `0.017`.

5 In the **z** text field, type `0.21`.

6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

7 In the **r** text field, type `0.017`.

8 In the **z** text field, type `Hvdom*0.98`.

Line Segment 3 (ls3)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **r** text field, type `0.05`.

5 In the **z** text field, type `0.05`.

6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

7 In the **r** text field, type `Wvdom*0.97`.

8 In the **z** text field, type `0.05`.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **pt3**, select Point 1 only.

5 On the object **pt4**, select Point 1 only.

6 Click **Build All Objects**.

Form Union (fin)

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

Merge Vertices 1 (mrv1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Merge Vertices**.

2 In the **Settings** window for **Merge Vertices**, locate the **Vertex to Keep** section.

3 Click to select the **Activate Selection** toggle button for **Vertex to keep**.

4 On the object **fin**, select Point 25 only.

5 Locate the **Vertex to Remove** section. Click to select the **Activate Selection** toggle button for **Vertex to remove**.

6 On the object **fin**, select Point 26 only.

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

Mesh Control Edges 1 (mce1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Edges**.

2 On the object **mrv1**, select Boundaries 10, 24, and 26 only.

Beverage

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

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

3 On the object **mce1**, select Domain 3 only.

Glass

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

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

3 On the object **mce1**, select Domain 2 only.

Vapor/Air

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

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

3 On the object **mce1**, select Domain 4 only.

Table

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

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

3 On the object **mce1**, select Domain 1 only.

Vapor-Liquid Surface

1. In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2. In the **Settings** window for **Explicit Selection**, type `Vapor-Liquid Surface` in the **Label** text field.
3. Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.
4. On the object **mce1**, select Boundary 8 only.
5. In the **Geometry** toolbar, click **Build All**.