Created in COMSOL Multiphysics 6.2

Free Convection in a Water Glass

Introduction

This example treats free convection in a glass of water. Free convection is a phenomenon that is often disregarded in chemical equipment. Yet, in certain circumstances it can be of great importance — for example, in fermentation processes, casting, and biochemical reactors. Natural convection can also be the leading contributor to transport in small reactors.

Model Definition

This example considers free convection in a glass of cold water at room temperature. You model the flow using the Nonisothermal Flow interface. The aim of this tutorial is to compute the flow pattern and the temperature distribution.

Initially, the glass and the water are both at 5°C, as if they had been taken directly from a refrigerator. The surrounding air and table are held constant at 25 °C. The glass wall has a finite thickness with a specific thermal conductivity. Due to rotational symmetry, you can model the whole system in 2D, using an axisymmetric geometry. The geometry and model domain are shown in Figure 1 below.

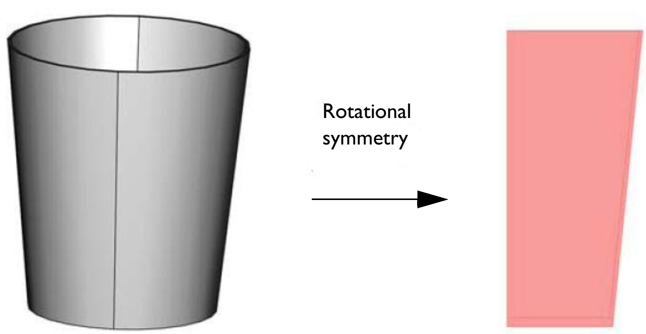


Figure 1: Geometry and computational domain.

The global mass and momentum balances for nonisothermal flow are coupled to an energy balance, where heat transport occurs through convection and conduction.

For the energy balance in the wall of the glass, only the conduction is considered. The thermal properties for the glass wall are assumed to be of silica glass.

BOUNDARY CONDITIONS

Assuming perfect contact between the table surface and the bottom of the glass, you can set the boundary condition to a temperature of 25°C. At the top and outer surfaces, use a convective heat flux boundary condition driven by the temperature difference between the glass and the surrounding atmosphere:

$$q = h(T_{\text{ext}} - T)$$

Here q is the inward heat flux and h is the heat transfer film coefficient. The Heat Transfer Module comes with a library of heat transfer coefficient functions (Ref. 1) that you can access easily and use in this application.

For the flow field, no slip conditions apply on the interior boundaries (between the glass and the water) while an axial symmetry condition applies on the axis of rotation and a slip condition on the open surface. In this case, the simulation runs for a period of 2 minutes.

Results and Discussion

The heat fluxes through the top surface, side wall and bottom of the glass are shown in Figure 2. Because of the low values of the heat transfer film coefficients, most of the heat is conducted to the water through the bottom boundary.

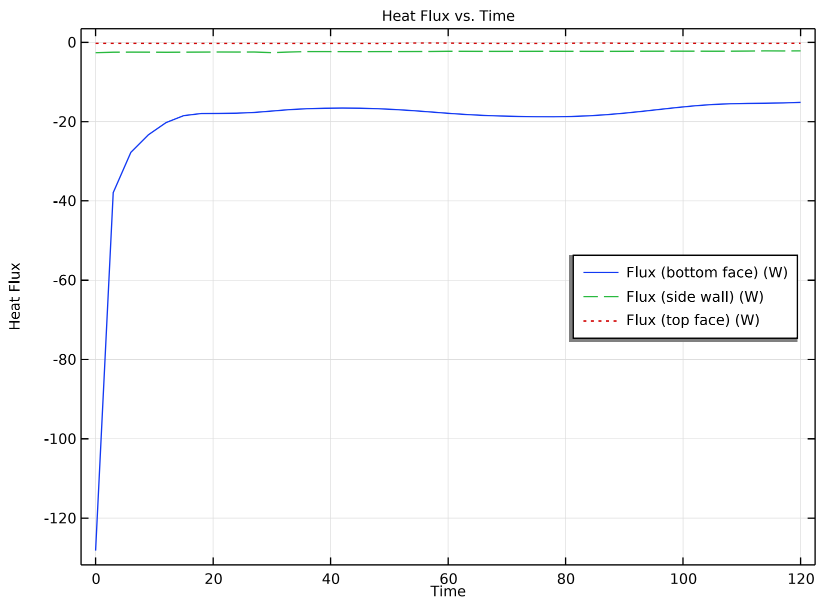


Figure 2: Heat flux through the top surface (dottet line), side wall (dashed line), and bottom of the glass (solid line).

When the fluid is heated at the bottom of the glass, the local density decreases, thereby inducing a flow inside the glass. Figure 3 shows temperature distributions for 30, 60, and 81s.

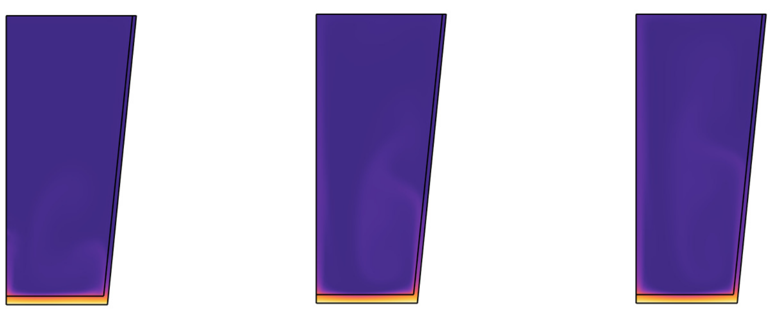


Figure 3: Temperature distribution at times 30, 60, and 81 s.

The buoyancy-driven flow induces recirculation zones in the glass. These recirculation zones are clearly seen in a streamline plot of the velocity field. Figure 4 shows the streamlines for the same output times as the previous figure.

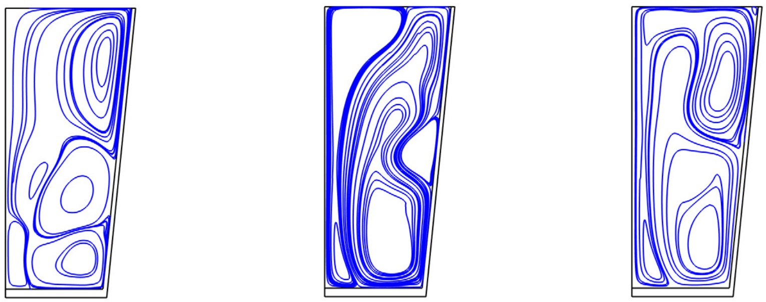


Figure 4: Velocity field at times 30, 60, and 81 s visualized with streamlines.

The following plot shows the temperature distribution in the glass after 2 minutes.

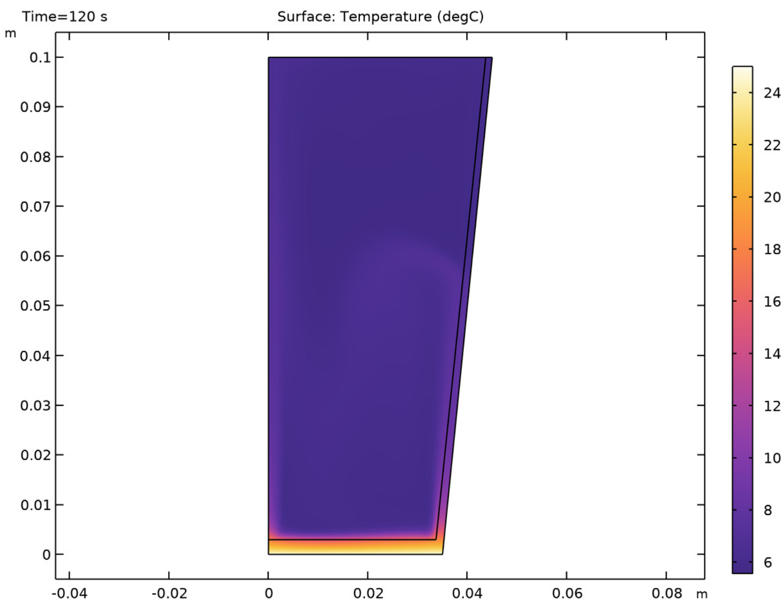


Figure 5: Temperature distribution at time 2 minutes.

Reference

1. A. Bejan, *Heat Transfer*, John Wiley & Sons, 1993.

Application Library path: `Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/cold_water_glass`

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 **Fluid Flow>Nonisothermal Flow>Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

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**
`r_top`	`4.5[cm]`	0.045 m	`Radius on the top`
`r_bottom`	`3.5[cm]`	0.035 m	`Radius at the bottom`
`Hg`	`10[cm]`	0.1 m	`Height of the glass`
`h_wall`	`0.13[cm]`	0.0013 m	`Thickness of the glass wall`
`h_bottom`	`0.3[cm]`	0.003 m	`Thickness of the bottom`

Name	Expression	Value	Description
Vlength	sqrt((r_top-r_bottom)^2+Hg^2)	0.1005 m	Length of the outer wall
rho0	1000[kg/m^3]	1000 kg/m³	Density reference

GEOMETRY 1

Polygon 1 (pol1)

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	h_bottom
r_bottom-h_wall	h_bottom
r_top-h_wall	Hg
0	Hg

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

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	Hg
0	0
r_bottom	0
r_top	Hg

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

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>Silica glass**.

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

5 In the tree, select **Built-in>Water, liquid**.

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

7 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 Select Domain 1 only.

Water, liquid (mat2)

1 In the **Model Builder** window, click **Water, liquid (mat2)**.

2 Select Domain 2 only.

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

4 Click **Create Selection**.

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

6 Click **OK**.

LAMINAR FLOW (SPF)

Set the flow as incompressible to ensure mass conservation, and then lock the pressure in the closed cavity to get a well-posed model. Use the Boussinesq approximation to account for the buoyancy force.

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 **Water**.

4 Locate the **Physical Model** section. From the **Compressibility** list, choose **Incompressible flow**.

5 Select the **Include gravity** check box.

6 Specify the $\mathbf{r}_{\mathrm{ref}}$ vector as

r_top-h_wall	r
Hg	z

7 Click to expand the **Discretization** section. From the **Discretization of fluids** list, choose **P2+P1**.

This setting gives quadratic elements for the velocity field.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 6 only.

Wall 2

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

2 Select Boundary 5 only.

3 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.

4 From the **Wall condition** list, choose **Slip**.

MULTIPHYSICS

Nonisothermal Flow 1 (nitf1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Nonisothermal Flow 1 (nitf1)**.

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

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

Set the ambient temperature to be used in boundary conditions of the Heat Transfer interface.

DEFINITIONS

Ambient Properties 1 (ampr1)

1 In the **Physics** toolbar, click **Shared Properties** and choose **Ambient Properties**.

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

3 In the T_{amb} text field, type `298.15[K]`.

HEAT TRANSFER IN FLUIDS (HT)

Initial Values 1

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

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

3 In the T text field, type `278.15[K]`.

Solid 1

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

2 Select Domain 1 only.

Temperature 1

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

2 Select Boundary 2 only.

Use the ambient temperature defined before in the **Ambient Settings** section of the interface.

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

4 From the T_0 list, choose **Ambient temperature (ampr1)**.

Heat Flux 1

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

2 Select Boundary 7 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 From the **Flux type** list, choose **Convective heat flux**.

5 From the **Heat transfer coefficient** list, choose **External natural convection**.

6 From the list, choose **Inclined wall**.

7 In the L text field, type `Vlength`.

8 In the φ text field, type `acos(Hg/Vlength)`.

For the external temperature and pressure, use the values defined in the **Ambient Settings** section of the interface.

9 From the T_{ext} list, choose **Ambient temperature (ampr1)**.

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

Heat Flux 2

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

2 Select Boundaries 5 and 8 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 From the **Flux type** list, choose **Convective heat flux**.

When the Rayleigh number is less than 10^4 for laminar flow, the conduction becomes far more dominant than the convection. Therefore, set the heat transfer coefficient to 2 W/(m^2·K) which corresponds to the thermal resistance within a small layer of air at 298.15 K that floats above the glass.

5 In the h text field, type 2.

For the external temperature, use the value defined before in the **Ambient Settings** section of the interface.

6 From the T_{ext} list, choose **Ambient temperature (ampr1)**.

MESH 1

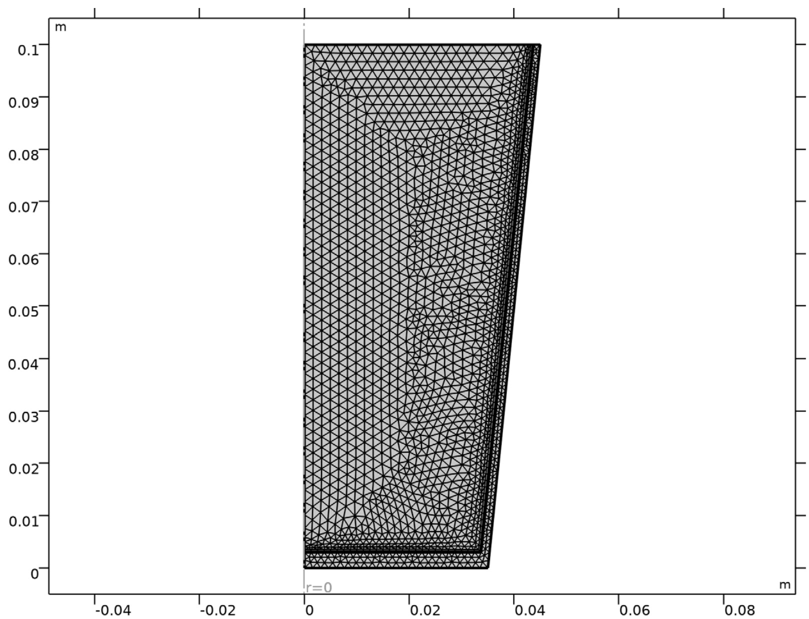
Use a physics-controlled mesh to get boundary layers at the water-glass interface.

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 **Fine**.

4 Click **Build All**.



STUDY 1

Step 1: Time Dependent

Since free convection is a rather slow phenomenon, run the problem for a period of 2 minutes.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,3[s],2[min])`.

There are a lot of secondary flow effects so you need to tighten the tolerance.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `1e-3`.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Absolute Tolerance** section.

4 From the **Tolerance method** list, choose **Manual**.

5 In the **Absolute tolerance** text field, type `2.5e-5`.

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

RESULTS

Velocity (spf)

The first default plot shows the velocity magnitude in a 2D plot, corresponding to one slice of the axisymmetric solution.

Pressure (spf)

The second default plot shows the pressure field in a 2D contour plot.

Temperature (ht)

The fourth default plot shows the 2D temperature distribution.

Surface 1

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

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

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

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

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

The plot in the **Graphics** window should look like that in Figure 5.

Temperature (ht)

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

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

3 From the **Time (s)** list, choose **30**.

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

Compare the result with the left plot in Figure 3.

Repeat the previous instruction for times 60 and 81 to generate the middle and right plots.

To produce the series of snapshots of the velocity streamlines shown in Figure 4, proceed with the following steps.

Velocity Streamlines, 2D

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

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

Streamline 1

1 In the **Velocity Streamlines, 2D** toolbar, click **Streamline**.

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

3 From the **Positioning** list, choose **Starting-point controlled**.

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

Velocity Streamlines, 2D

1 In the **Model Builder** window, click **Velocity Streamlines, 2D**.

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

3 From the **Time (s)** list, choose **30**.

4 In the **Velocity Streamlines, 2D** toolbar, click **Plot**.

Compare the result with the left plot in Figure 4.

Repeat the previous instruction for the times 60 and 81 to generate the middle and right plots.

Finally, compute and plot the fluxes through the top, bottom, and side wall of the glass.

Line Integration 1

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

2 Select Boundary 2 only.

3 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)>Heat Transfer in Fluids>Boundary fluxes>ht.ntflux - Normal total heat flux - W/m²**.

4 Replace the variable description by `Flux (bottom face)`.

5 Click **Evaluate**.

Line Integration 2

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

2 Select Boundary 7 only.

3 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)>Heat Transfer in Fluids>Boundary fluxes>ht.ntflux - Normal total heat flux - W/m²**.

4 Replace the variable description by `Flux (side wall)`.

5 Click **Evaluate**.

Line Integration 3

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

2 Select Boundaries 5 and 8 only.

3 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)>Heat Transfer in Fluids>Boundary fluxes>ht.ntflux - Normal total heat flux - W/m²**.

4 Replace the variable description by `Flux (top face)`.

5 Click **Evaluate**.

Heat Flux vs. Time

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

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

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

4 In the **Title** text area, type `Heat Flux vs. Time`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `Time`.

7 Select the **y-axis label** check box. In the associated text field, type `Heat Flux`.

8 Locate the **Grid** section. Select the **Manual spacing** check box.

9 In the **x spacing** text field, type 20.

10 In the **y spacing** text field, type 20.

Table Graph 1

1 Right-click **Heat Flux vs. Time** and choose **Table Graph**.

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

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

4 In the **Heat Flux vs. Time** toolbar, click **Plot**.

Heat Flux vs. Time

In the **Model Builder** window, click **Heat Flux vs. Time**.

Table Graph 2

1 In the **Heat Flux vs. Time** toolbar, click **Table Graph**.

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

3 From the **Table** list, choose **Table 2**.

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

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

6 In the **Heat Flux vs. Time** toolbar, click **Plot**.

Heat Flux vs. Time

1 In the **Model Builder** window, click **Heat Flux vs. Time**.

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

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

Table Graph 3

1 In the **Heat Flux vs. Time** toolbar, click **Table Graph**.

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

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

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

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

6 In the **Heat Flux vs. Time** toolbar, click **Plot**.

Compare the resulting plot with that in Figure 2.