

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

- I 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 \bigcirc Study.
- 5 In the Select Study tree, select General Studies>Time Dependent.
- 6 Click **M** Done.

GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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 I

Polygon I (poll)

I 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)

I 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

I 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 (mat I)

- I In the Model Builder window, under Component I (compl)>Materials click Silica glass (matl).
- **2** Select Domain 1 only.

Water, liquid (mat2)

- I 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 here a 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.

- I In the Model Builder window, under Component I (compl) 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}_{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 I

- I In the Physics toolbar, click 💮 Points and choose Pressure Point Constraint.
- 2 Select Point 6 only.

Wall 2

- I 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)

- I In the Model Builder window, under Component I (compl)>Multiphysics click Nonisothermal Flow I (nitfl).
- 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)

- I 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

- I In the Model Builder window, under Component I (compl)>Heat Transfer in Fluids (ht) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the *T* text field, type 278.15[K].

Solid I

- I In the Physics toolbar, click **Domains** and choose Solid.
- **2** Select Domain 1 only.

Temperature I

- I 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 (amprl)**.

Heat Flux 1

- I 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 (amprl).

IO From the p_A list, choose **Ambient absolute pressure (amprl)**.

Heat Flux 2

- I 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 $2 \text{ W/(m}^2 \cdot \text{K})$ 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 (amprl)**.

MESH I

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

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 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 I

Step 1: Time Dependent

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

- I In the Model Builder window, under Study I click Step I: 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 (soll)

- I In the Study toolbar, click **here** Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node, then click Time-Dependent Solver I.
- **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.

```
Isothermal Contours (ht)
```

This default plot shows the isothermal contours of the temperature field in the glass. To reproduce the three snapshots in Figure 3, do as follows.

Temperature, 2D

- I In the Home toolbar, click 🚛 Add Plot Group and choose 2D Plot Group.
- 2 In the Settings window for 2D Plot Group, type Temperature, 2D in the Label text field.

Surface 1

- I In the Temperature, 2D toolbar, click Surface.
- 2 In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
 Heat Transfer in Fluids>Temperature>T Temperature K.
- **3** Locate the **Expression** section. From the **Unit** list, choose **degC**.

- 4 Locate the Coloring and Style section. Click Change Color Table.
- 5 In the Color Table dialog box, select Thermal>HeatCameraLight in the tree.
- 6 Click OK.
- 7 In the Temperature, 2D toolbar, click 💿 Plot.
- 8 Click the $4 \rightarrow$ Zoom Extents button in the Graphics toolbar.

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

Temperature, 2D

- I In the Model Builder window, click Temperature, 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 Temperature, 2D 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

- I 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

- I 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

- I 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 **O 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

- I 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 I (compl)> 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

- I 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 I (compl)> 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

- I 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 I (compl)> 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

- I In the **Results** toolbar, click \sim **ID Plot Group**.
- 2 In the Settings window for ID 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.
- **IO** In the **y spacing** text field, type 20.

Table Graph 1

- I 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 **O Plot**.

Heat Flux vs. Time

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

Table Graph 2

- I In the Heat Flux vs. Time toolbar, click I 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

- I In the Model Builder window, click Heat Flux vs. Time.
- 2 In the Settings window for ID Plot Group, locate the Legend section.
- **3** From the **Position** list, choose **Middle right**.

Table Graph 3

- I In the Heat Flux vs. Time toolbar, click III 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.