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 Non-Isothermal 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.

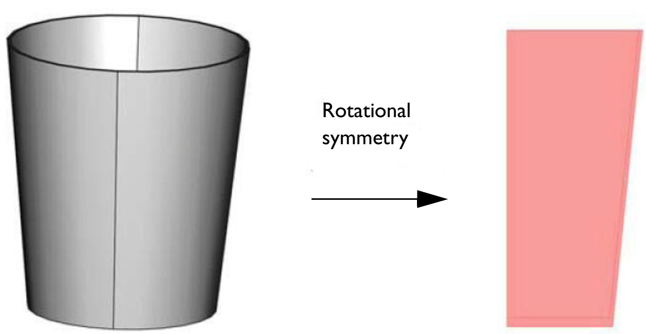


Figure 1: Geometry and computational domain.

The global mass and momentum balances for non-isothermal 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:

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

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.

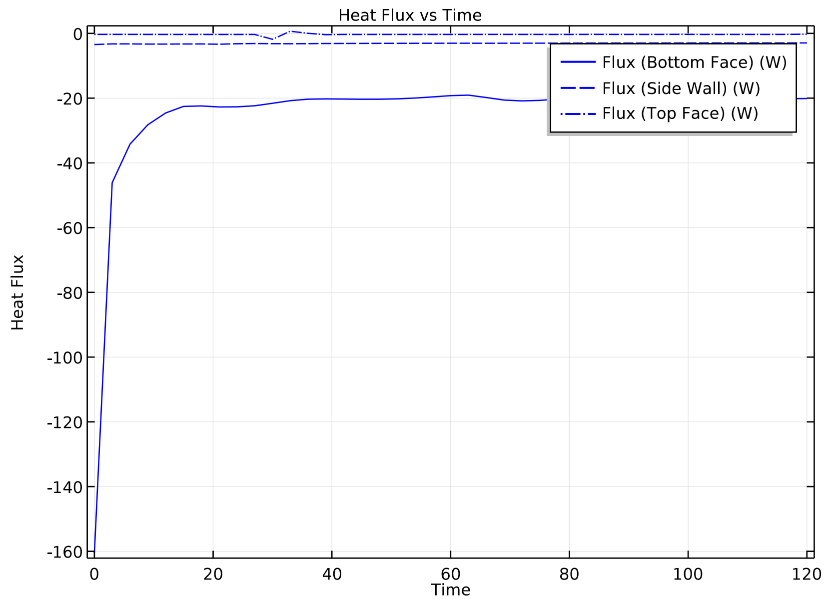


Figure 2: Heat flux through the top surface (solid line), side wall (dotted line), and bottom of the glass (dashed line).

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 equally spaced time intervals.

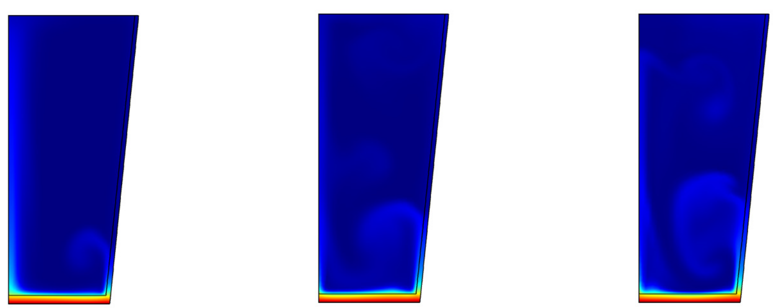


Figure 3: Temperature distribution at t = 30, 60, and 81 s.

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.

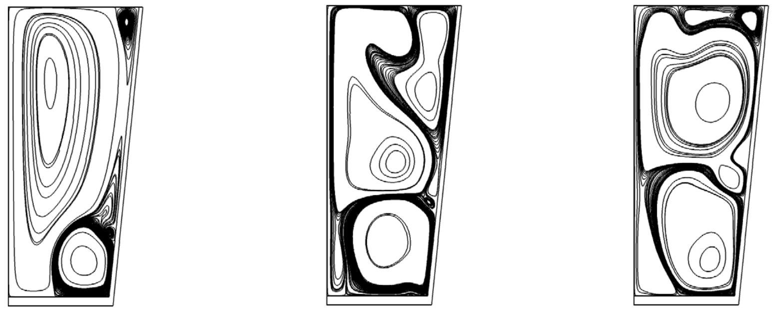


Figure 4: Velocity field after t = 30, 60, and 81 s visualized with streamlines.

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

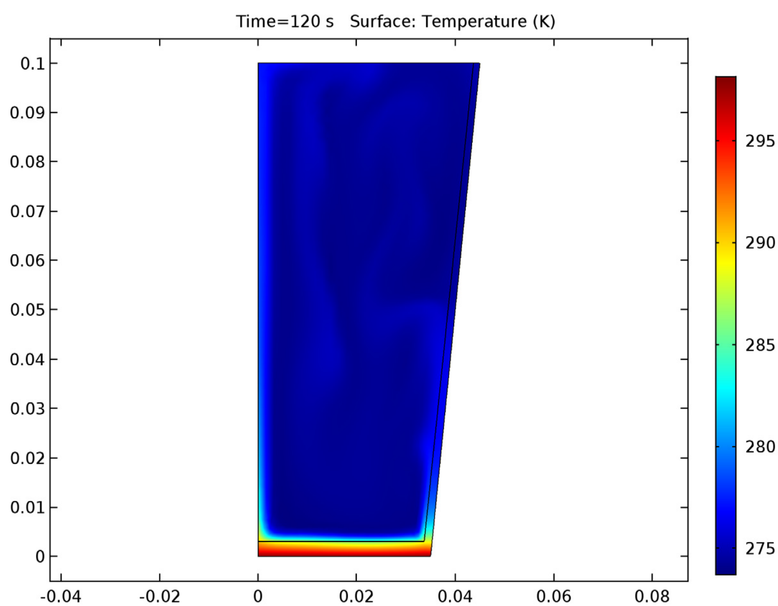


Figure 5: Temperature distribution after 2 minutes.

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

1 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>Non-Isothermal Flow>Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select study** tree, select **Preset Studies for Selected Physics Interfaces>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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 On the **Geometry** toolbar, click **Primitives** and choose **Polygon**.

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

3 In the **r** text field, type `0 r_bottom-h_wall r_top-h_wall 0`.

4 In the **z** text field, type `h_bottom h_bottom Hg Hg`.

5 Click the **Build All Objects** button.

Polygon 2 (pol2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Polygon**.

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

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

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

5 Click the **Build All Objects** button.

ADD MATERIAL

1 On 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.

MATERIALS

Silica glass (mat1)

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

2 Select Domain 1 only.

ADD MATERIAL

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

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

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

MATERIALS

Water, liquid (mat2)

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

2 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat2)**.

3 Select Domain 2 only.

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

5 Click **Create Selection**.

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

7 Click **OK**.

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 **Water**.

4 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

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

This setting gives quadratic elements for the velocity field.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Initial Values 1**.

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

3 In the p text field, type `rho0*g_const*(0.1[m]-z)`.

This setting gives an initial pressure field consistent with the volume force. The initial pressure field must be also consistent with the pressure constraint (see the Pressure Point Constraint 1 section below).

Volume Force 1

1 On the **Physics** toolbar, click **Domains** and choose **Volume Force**.

2 Select Domain 2 only.

3 In the **Settings** window for Volume Force, locate the **Volume Force** section.

4 Specify the **F** vector as

`0`	r
`-g_const*nitf1.rho`	z

Wall 2

1 On 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 **Boundary condition** list, choose **Slip**.

Because this is a closed cavity flow, lock the pressure.

Pressure Point Constraint 1

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

2 Select Point 6 only.

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 `273.15[K]`.

4 In the **Model Builder** window, click **Heat Transfer in Fluids (ht)**.

Heat Transfer in Solids 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Transfer in Solids**.

2 Select Domain 1 only.

Temperature 1

1 On 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 `298.15[K]`.

Heat Flux 1

1 On 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 Click the **Convective heat flux** button.

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)`.

9 In the T_{ext} text field, type `298.15[K]`.

Heat Flux 2

1 On 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 Click the **Convective heat flux** button.

When the Rayleigh number is lesser than 10^4 for laminar flow, the conduction becomes far more dominant than the convection losses. Therefore, set the heat transfer coefficient to 2 $W/(m^2{\cdot}K)$ 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`.

6 In the T_{ext} text field, type `298.15[K]`.

MESH 1

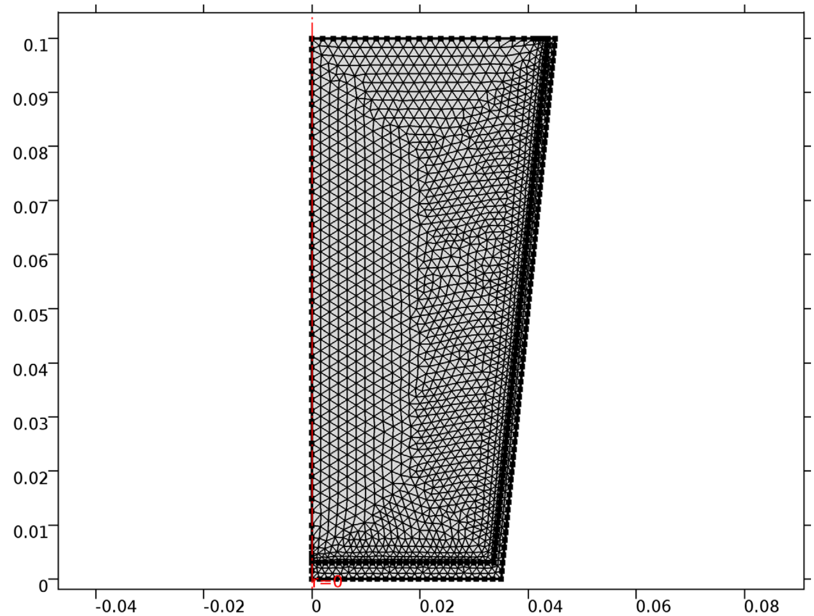
Use a physics-controlled mesh to get boundary layers at the water-glass interface.

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

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

3 From the **Element size** list, choose **Fine**.

4 Click the **Build All** button.



STUDY 1

Step 1: Time Dependent

Because 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 **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. Select the **Relative tolerance** check box.
5. In the associated text field, type `1e-3`.

Solution 1 (sol1)

1. On 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 Locate the **Absolute Tolerance** section. In the **Tolerance** text field, type `2.5e-5`.

5 On 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:

2D Plot Group 6

1 On 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.

Temperature, 2D

1 Right-click **Temperature, 2D** and choose **Surface**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Heat Transfer in Fluids>Temperature>T - Temperature**.

3 On the **Temperature, 2D** toolbar, click **Plot**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.

The plot in the Graphics window should resemble that in Figure 5.

5 In the **Model Builder** window, click **Temperature, 2D**.

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

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

8 On the **Temperature, 2D** toolbar, click **Plot**.

Compare the result with the left plot in Figure 3.

Repeat the previous instruction for the 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:

2D Plot Group 7

1 On 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.

Velocity Streamlines, 2D

1 Right-click **Velocity Streamlines, 2D** and choose **Streamline**.

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

3 From the **Positioning** list, choose **Start point controlled**.

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

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

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

7 On 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 trough the top, bottom, and side wall of the glass.

Line Integration 1

On the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.

Derived Values

1 Select Boundary 2 only.

2 In the **Settings** window for Line Integration, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Heat Transfer in Fluids>Boundary fluxes>ht.ntflux - Normal total heat flux**.

3 Locate the **Expression** section. Select the **Description** check box.

4 In the associated text field, type `Flux (Bottom Face)`.
5 Locate the **Integration Settings** section. Select the **Compute surface integral** check box.
6 Click the **Evaluate** button.

TABLE

Go to the **Table** window.

RESULTS

Line Integration 2

On the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.

Derived Values

1 Select Boundary 7 only.
2 In the **Settings** window for Line Integration, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Heat Transfer in Fluids>Boundary fluxes>ht.ntflux - Normal total heat flux**.
3 Locate the **Expression** section. Select the **Description** check box.
4 In the associated text field, type `Flux (Side Wall)`.
5 Locate the **Integration Settings** section. Select the **Compute surface integral** check box.
6 Click the **Evaluate** button.

TABLE

Go to the **Table** window.

RESULTS

Line Integration 3

On the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.

Derived Values

1 Select Boundaries 5 and 8 only.
2 In the **Settings** window for Line Integration, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Heat Transfer in Fluids>Boundary fluxes>ht.ntflux - Normal total heat flux**.

3 Locate the **Expression** section. Select the **Description** check box.

4 In the associated text field, type `Flux (Top Face)`.

5 Locate the **Integration Settings** section. Select the **Compute surface integral** check box.

6 Click the **Evaluate** button.

TABLE

Go to the **Table** window.

RESULTS

1D Plot Group 8

1 On 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. Select the **x-axis label** check box.

6 In the associated text field, type `Time`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Heat Flux`.

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

10 In the **x spacing** text field, type `20`.

11 In the **y spacing** text field, type `20`.

Table Graph 1

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

Heat Flux vs Time

1 In the **Settings** window for Table Graph, locate the **Coloring and Style** section.

2 From the **Color** list, choose **Blue**.

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

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

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

Table Graph 2

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

Heat Flux vs Time

1 In the **Settings** window for Table Graph, locate the **Data** section.
2 From the **Table** list, choose **Table 2**.
3 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
4 From the **Color** list, choose **Blue**.
5 Locate the **Legends** section. Select the **Show legends** check box.
6 On the **Heat Flux vs Time** toolbar, click **Plot**.
7 In the **Model Builder** window, click **Heat Flux vs Time**.

Table Graph 3

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

Heat Flux vs Time

1 In the **Settings** window for Table Graph, locate the **Data** section.
2 From the **Table** list, choose **Table 3**.
3 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dash-dot**.
4 From the **Color** list, choose **Blue**.
5 Locate the **Legends** section. Select the **Show legends** check box.
6 On the **Heat Flux vs Time** toolbar, click **Plot**.

   Compare the resulting plot with that in Figure 2.