



Buoyancy Flow of Free Fluids

Introduction

This example follows the benchmark buoyancy flow posed by de Vahl Davis ([Ref. 1](#)) for free fluids. Buoyancy flow of free fluids is very important in Earth sciences with temperature and concentration affecting density in moving fluids, for example, in pipes, along shorelines, and within lakes. Here the buoyancy flow results from density that varies with a temperature change. The COMSOL Multiphysics results match those from the published study ([Ref. 1](#)). The model was provided by John Kamel of the University of Notre Dame.

This example repeatedly solves a problem of buoyant flow in a square cavity. It thereby analyzes different temperature distributions and convective flow patterns from variations in, for example, fluid properties, cavity size, and temperature drops. The iterative process is tuned for a fast, efficient solution using nondimensional parameters and a Boussinesq term for the buoyant drive with the Laminar Flow and the Heat Transfer in Fluids interfaces. As an alternative, the same problem can be solved using only the Nonisothermal Flow interface available with the CFD Module or the Heat Transfer Module.

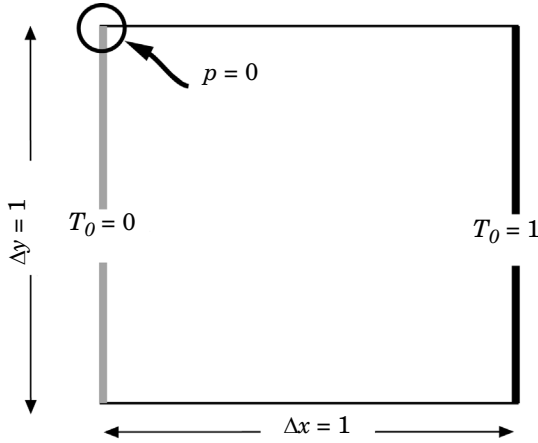


Figure 1: Domain geometry and boundary conditions for the heat balance in this example of buoyant flow in free fluids.

Model Definition

The previous figure illustrates the model geometry. The fluid fills a square cavity with impermeable walls so the fluid flows freely within it but does not exit from it. The right and left edges of the cavity are, respectively, the high and low temperature sources. The

upper and lower boundaries are insulated. The temperature differential produces the density variation that drives the buoyant flow.

The laminar flow and heat transfer interfaces in this example are bidirectionally coupled. The Boussinesq term defines a force dependent on the temperature and the fluid velocity transports heat.

It is possible to formulate the incompressible Navier–Stokes equations with a Boussinesq buoyancy term on the right-hand side to account for the lifting force due to thermal expansion:

$$\rho_0(\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p + \nabla \cdot \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) + \rho_0 \mathbf{g} \alpha_p (T - T_0)$$

$$\nabla \cdot \mathbf{u} = 0$$

In these expressions, the dependent variables for flow are \mathbf{u} , the vector of fluid velocity, and pressure, p . T represents temperature, T_0 is a reference temperature, \mathbf{g} denotes gravity acceleration, ρ_0 gives the reference density, μ is the dynamic viscosity, and α_p equals the coefficient of volumetric thermal expansion.

The heat balance comes from the conduction-convection equation

$$\rho_0 C_p \mathbf{u} \cdot \nabla T - \nabla \cdot (k \nabla T) = 0$$

where k denotes the thermal conductivity, and C_p is the specific heat capacity of the fluid.

For the Navier–Stokes equations, impermeable, no-slip boundary conditions apply. The no-slip condition results in zero velocity at the wall, with pressure within the domain remaining undefined. Because the lack of information about p makes it difficult to achieve convergence, you arbitrarily fix the pressure at a point.

The boundary conditions for the heat transfer interface are the fixed high and low temperatures on the vertical walls, with insulation conditions elsewhere, as shown in [Figure 1](#).

Notes About the COMSOL Implementation

The model addresses a wide range of cavity sizes, fluid properties, and temperature drops using material properties set up with nondimensional Rayleigh and Prandtl numbers. The Rayleigh number, $Ra = (C_p \rho^2 g \alpha_p T L^3) / (\mu k)$, gives the ratio of buoyant to viscous forces. Here L is the length of a side wall. The Prandtl number, $Pr = (\mu C_p / k)$, gives the ratio of kinematic viscosity to thermal diffusivity.

Setting the body force in the y -direction for the momentum equation to $F_y = (\text{Ra}/\text{Pr})(T - T_c)$, and the fluid properties to $C_p = \text{Pr}$, and $\rho = \mu = k = 1$, produces a set of equations with nondimensional variables p , \mathbf{u} , and T .

As Ra increases, viscous forces decrease in importance. You can examine a wide range of scenarios sequencing through different values of Ra using an **Auxiliary sweep** with continuation selected for the Ra parameter in the **Stationary** study step. At high Ra values, starting with a good initial condition and a well-tuned mesh becomes increasingly important. Because the continuation feature in the parametric solver uses extrapolation from the previous solution as the initial condition for the next one, together with automatic step length control, it fulfills the first of these two requirements. Then getting a well-tuned mesh is straightforward: simply set up the mesh for the most difficult problem to solve — the one with the highest value of Ra . To that end, the element size near the prescribed temperature boundaries corresponds to the thickness of the boundary layer when $\text{Ra} = 10^6$.

Results and Discussion

As noted earlier, this COMSOL Multiphysics example reproduces a benchmark buoyant flow problem reported in [Ref. 1](#). The composite images in [Figure 2](#) summarize temperatures (surface), velocity fields (arrows), and x -velocities (contours) for four Ra values. With the given definitions for the dimensionless variable, it follows that as Ra increases so does temperature. The results in the figure indicate how the vigor and complexity of the convection increase at higher values of Ra , as expected. These results are virtually identical to the benchmark solution except the COMSOL Multiphysics plots provide higher resolution than the images in the original publication.

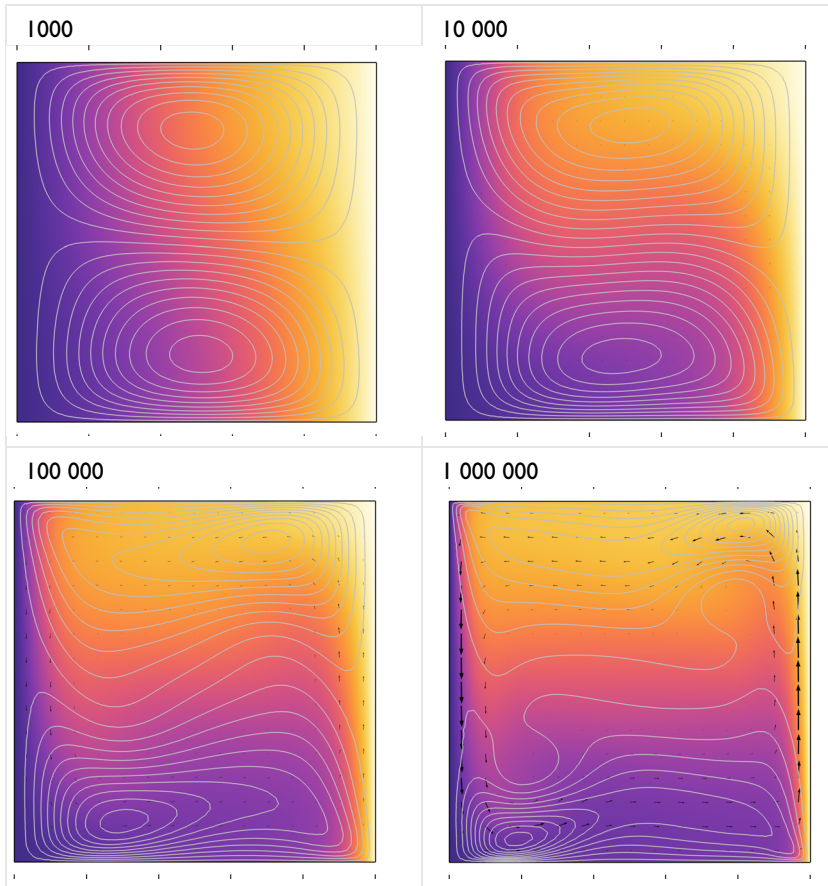


Figure 2: Dimensionless solution for buoyancy flow in a fluid-filled cavity at increasing Rayleigh number: Temperature (surface plot), velocity field (arrows), and x-velocity (contours). These results from the COMSOL Multiphysics simulation are in excellent agreement with the results published in [Ref. 1](#).

The COMSOL Multiphysics model described here represents the buoyant drive with a Boussinesq term. As an alternative, you could use the Nonisothermal Flow interface provided with the laminar flow interface. Using the Boussinesq approach, however, demonstrates a well-established method for reducing computational effort while still representing buoyant flow.

References


1. G. de Vahl Davis and I.P. Jones, “Natural Convection in a Square Cavity: A Comparison Exercise,” *Int. J. Num. Meth. in Fluids*, vol. 3, pp. 227–248, 1983.
2. G. de Vahl Davis, “Natural Convection of Air in a Square Cavity: A Bench Mark Numerical Solution,” *Int. J. Num. Meth. in Fluids*, vol. 3, pp. 249–264, 1983.

Application Library path: COMSOL_Multiphysics/Fluid_Dynamics/buoyancy_free




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**.
- 2 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.
- 3 Click **Add**.
- 4 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.
- 5 Click **Add**.
- 6 Click  **Study**.
- 7 In the **Select Study** tree, select **General Studies>Stationary**.
- 8 Click  **Done**.

GLOBAL DEFINITIONS

Parameters I




- 1 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
Tc	0	0	Low temperature
Th	1	1	High temperature
Pr	0.71	0.71	Prandtl number
p0	0	0	Reference pressure
Ra	0	0	Rayleigh number

GEOMETRY I

Square I (sq1)

- 1 In the **Geometry** toolbar, click  **Square**.
- 2 In the **Settings** window for **Square**, click  **Build All Objects**.
- 3 Click the  **Zoom Extents** button in the **Graphics** toolbar.

LAMINAR FLOW (SPF)


Fluid Properties I

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Fluid Properties 1**.
- 2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.
- 3 From the p list, choose **User defined**. In the associated text field, type 1.
- 4 From the μ list, choose **User defined**. In the associated text field, type 1.

Initial Values I


- 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 p text field, type p0.

Volume Force I

- 1 In the **Physics** toolbar, click  **Domains** and choose **Volume Force**.
- 2 In the **Settings** window for **Volume Force**, locate the **Volume Force** section.
- 3 Specify the \mathbf{F} vector as

0	x
$(Ra/Pr) * (T - T_c)$	y

Pressure Point Constraint 1

- 1 In the **Physics** toolbar, click  **Points** and choose **Pressure Point Constraint**.
- 2 Select Point 2 only.
- 3 In the **Settings** window for **Pressure Point Constraint**, locate the **Pressure Constraint** section.
- 4 In the p_0 text field, type p_0 .

Volume Force 1

- 1 In the **Model Builder** window, click **Volume Force 1**.
- 2 In the **Settings** window for **Volume Force**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **All domains**.

HEAT TRANSFER IN FLUIDS (HT)


Fluid 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Fluids (ht)** click **Fluid 1**.
- 2 In the **Settings** window for **Fluid**, locate the **Model Input** section.
- 3 From the p_A list, choose **User defined**. In the associated text field, type p_0 .
- 4 Locate the **Heat Convection** section. From the \mathbf{u} list, choose **Velocity field (spf)**.
- 5 Locate the **Heat Conduction, Fluid** section. From the k list, choose **User defined**. In the associated text field, type 1.
- 6 Locate the **Thermodynamics, Fluid** section. From the **Fluid type** list, choose **Gas/Liquid**.
- 7 From the p list, choose **User defined**. In the associated text field, type 1.
- 8 From the γ list, choose **User defined**. From the C_p list, choose **User defined**. In the associated text field, type Pr .

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 T text field, type T_c .

Temperature 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Temperature**.
- 2 Select Boundary 1 only.
- 3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

- 4 In the T_0 text field, type T_c .


Temperature 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Temperature**.
- 2 Select Boundary 4 only.
- 3 In the **Settings** window for **Temperature**, locate the **Temperature** section.
- 4 In the T_0 text field, type T_h .

MESH 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
- 2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
- 3 From the **Element size** list, choose **Extra fine**.

Edge 1

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Edge**.
- 2 Select Boundaries 1 and 4 only.

Size 1

- 1 Right-click **Edge 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Extremely fine**.


Free Triangular 1

- 1 In the **Mesh** toolbar, click  **Free Triangular**.
- 2 In the **Settings** window for **Free Triangular**, click  **Build All**.

STUDY 1

Step 1: Stationary

Set up an auxiliary continuation sweep for the Ra parameter.

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.
- 3 Select the **Auxiliary sweep** check box.
- 4 Click  **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list
Ra (Rayleigh number)	1 1e1 1e2 1e3 1e4 1e5 1e6

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

RESULTS

Velocity (spf)

The first default plot group shows the velocity magnitude. Notice the high velocities near the lateral walls due to buoyancy effects.


Pressure (spf)

The second default plot group shows the pressure distribution. Because the cavity is closed, the pressure distribution is solely due to gravity.

Temperature (ht)

The third default plot group shows the temperature field as a surface plot. Add arrow and contour plots of the velocity field.

Arrow Surface 1


- 1 In the **Model Builder** window, expand the **Results>Temperature (ht)** node.
- 2 Right-click **Temperature (ht)** and choose **Arrow Surface**.
- 3 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.
- 4 Select the **Scale factor** check box. In the associated text field, type $2e-4$.
- 5 From the **Color** list, choose **Black**.
- 6 In the **Temperature (ht)** toolbar, click  **Plot**.

Contour 1

- 1 In the **Model Builder** window, right-click **Temperature (ht)** and choose **Contour**.
- 2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Laminar Flow>Velocity and pressure>Velocity field - m/s>u - Velocity field, x-component**.
- 3 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 4 From the **Color** list, choose **Gray**.
- 5 Clear the **Color legend** check box.

Buoyancy

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

- 2** In the **Settings** window for **2D Plot Group**, type Buoyancy in the **Label** text field.
- 3** Locate the **Data** section. From the **Parameter value (Ra)** list, choose **1000**.
- 4** In the **Buoyancy** toolbar, click  **Plot**.

Repeat the previous steps for the parameters **le4**, **le5**, and **le6** to reproduce the remaining plots in [Figure 2](#).

