Created in COMSOL Multiphysics 6.1

Buoyancy Flow in Water

Introduction

This example studies the stationary state of free convection in a cavity filled with water and bounded by two vertical plates. To generate the buoyancy flow, the plates are heated at different temperatures, bringing the regime close to the transition between laminar and turbulent flow.

An important step in setting up a convection model is to assess whether the flow stays laminar or becomes turbulent. It is also important to approximate how fine should be the mesh needed to resolve velocity and temperature gradients. Both of these approximations rely on the velocity scale of the model. This makes the setup of natural convection problems challenging because the resulting velocity is part of the nonlinear solution. There are, however, tools to approximate scales for natural convection problems. These tools are demonstrated in this application using simple 2D and 3D geometries.

A first 2D model representing a square cavity (see Figure 1) focuses on the convective flow.

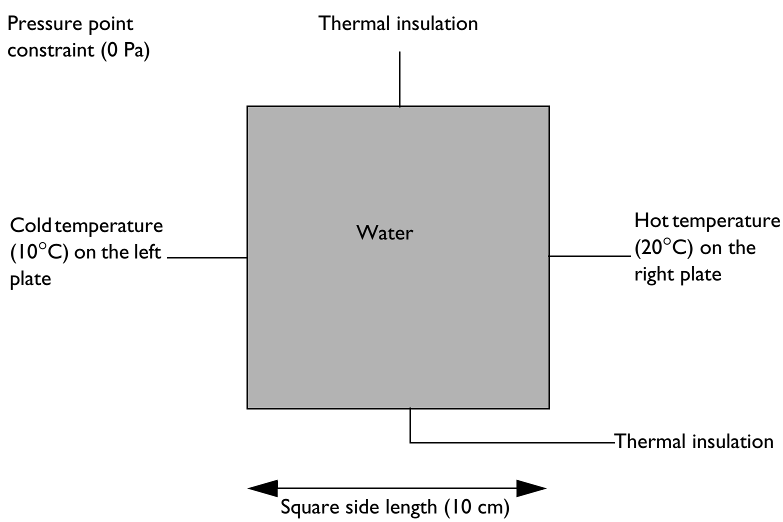


Figure 1: Domain geometry and boundary conditions for the 2D model (square cavity).

The 3D model (see Figure 2) extends the geometry to a cube. Compared to the 2D model, the front and back sides are additional boundaries that may influence the fluid behavior.

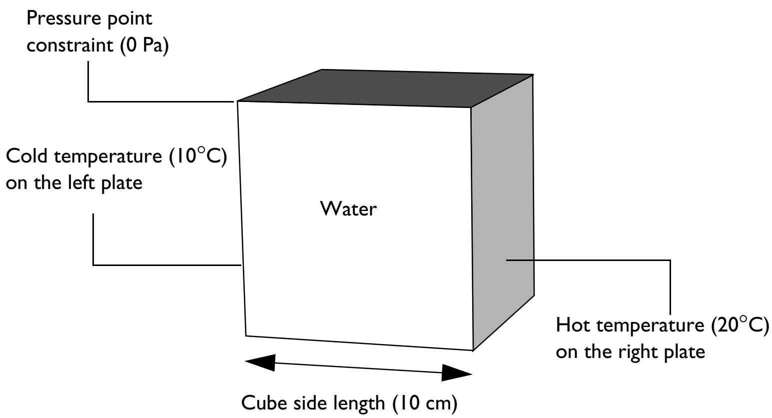


Figure 2: Domain geometry and boundary conditions for the 3D model (cubic cavity).

Both models calculate and compare the velocity field and the temperature field. The predefined Nonisothermal Flow interface available in the Heat Transfer Module provides appropriate tools to fully couple the heat transfer and the fluid dynamics.

Model Definition

2D MODEL

Figure 1 illustrates the 2D model geometry. The fluid fills a square cavity with impermeable walls, so the fluid flows freely within the cavity but cannot leak out. The right and left edges of the cavity are maintained at high and low temperatures, respectively. The upper and lower boundaries are insulated. The temperature differential produces the density variation that drives the buoyant flow.

The compressible Navier-Stokes equation contains a buoyancy term on the right-hand side to account for the lifting force due to thermal expansion that causes the density variations:

$$\rho(\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p + \nabla \cdot \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^{T}) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u}) + \rho \mathbf{g}$$

In this expression, the dependent variables corresponding to flow are the fluid velocity vector, $\mathbf{u}$, and the pressure, p. The constant $\mathbf{g}$ denotes the gravitational acceleration, ρ gives the temperature-dependent density, and μ is the temperature-dependent dynamic viscosity.

As the density changes are relatively small in this model, it is possible to use a simplified version of Navier-Stokes equation based on Boussinesq approximation:

$$\rho_{\mathrm{ref}}(\mathbf{u}\cdot\nabla)\mathbf{u} = -\nabla p + \nabla\cdot\mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^{\mathrm{T}}) + (1-\alpha_p(T-T_{\mathrm{ref}}))\mathbf{g}$$

where T_{ref} is the reference temperature, ρ_{ref} is the reference density, and α_p is the isobaric compressibility coefficient evaluated at the reference pressure and temperature.

Because the model only contains information about the pressure gradient, it estimates the pressure field up to a constant. To define this constant, you arbitrarily fix the pressure at a point. No slip boundary conditions apply on all boundaries. The no slip condition results in zero velocity at the wall but does not set any constraint on p.

At steady-state, the heat balance for a fluid reduces to the following equation:

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

Here T represents the temperature, k denotes the thermal conductivity, and C_p is the specific heat capacity of the fluid.

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.

3D MODEL

Figure 2 shows the geometry and boundary conditions of the 3D model. The cavity is now a cube with high and low temperatures applied respectively at the right and left surfaces. The remaining boundaries (top, bottom, front, and back) are thermally insulated.

MODEL ANALYSIS

Before starting the simulations, it is recommended to estimate the flow regime. To this end, four indicators are presented: the Reynolds, Grashof, Rayleigh, and Prandtl numbers. They are calculated using the thermophysical properties of water listed in Table 1, found in the Water, liquid material of the Built-in material library. The thermophysical properties are given at 283 K which is in the range of the temperatures observed in the model.

TABLE 1: THERMOPHYSICAL PROPERTIES FOR WATER AT 283 K.

PARAMETER	DESCRIPTION	VALUE
ρ	Density	$1.0 \cdot 10^3$ kg/m^3
μ	Dynamic viscosity	$1.3 \cdot 10^{-3}$ N·s/m^2
k	Thermal conductivity	0.58 W/(m·K)
C_p	Heat capacity at constant pressure	4.2 kJ/(kg·K)
α_p	Coefficient of thermal expansion	$9 \cdot 10^{-5}$ K^{-1}

Prandtl Number

Definition The Prandtl number is the ratio of fluid viscosity to thermal diffusivity. It is defined by:

$$\mathrm{Pr} = \frac{\mu C_p}{k}$$

At 283 K, $\mathrm{Pr} = 0.72$ for air and $\mathrm{Pr} = 9.4$ for water, as given in Ref. 2.

Boundary Layers The Prandtl number also indicates the relative thickness of the outer boundary layer, δ, and the thermal boundary layer, δ_T. In the present case, it is reasonable to estimate the ratio δ / δ_T by the relation (7.32 in Ref. 2)

$$\frac{\delta}{\delta_T} = \sqrt{\mathrm{Pr}} \tag{1}$$

The outer boundary layer is the distance from the wall to the region where the fluid stabilizes. It is different from the momentum boundary layer, δ_M, which measures the distance from the wall to the velocity peak.

Application in this Tutorial With the values given in Table 1, the Prandtl number for water at 283 K, is found to be of the order 1 or 10. According to Equation 1, δ and δ_T should then be of same order of magnitude.

Reynolds Number

Definition The Reynolds number estimates the ratio of inertial forces to viscous forces. It is defined by the formula

$$\mathrm{Re} = \frac{\rho UL}{\mu}$$

where U denotes the typical velocity and L the typical length.

At atmospheric pressure and at 283 K, air and water have the following properties (Ref. 2).

TABLE 2: THERMOPHYSICAL PROPERTIES OF AIR AND WATER AT 283K AND ATMOSPHERIC PRESSURE.

Parameter	Description	Air	Water
ρ	Density	1.25 kg/m^3	1000 kg/m^3
μ	Dynamic viscosity	$1.76\cdot10^{-5}$ N·s/m^2	$1.3\cdot10^{-3}$ N·s/m^2

You can thus approximate the Reynolds number by:

$$\mathrm{Re}_{\mathrm{air}} \approx 10^5 UL \qquad \mathrm{Re}_{\mathrm{water}} \approx 10^6 UL$$

In these relations, U has to be in meters per second and L in meters.

The Reynolds number can be rewritten as the velocity ratio

$$\mathrm{Re} = \frac{U}{\mu/(\rho L)}$$

which compares U to $\mu/(\rho L)$. The latter quantity is homogeneous to a velocity and can be seen as the typical velocity due to viscous forces.

Flow Regime The value of the Reynolds number is used to predict the flow regime. Generally, low values of Re correspond to laminar flow and high values to turbulent flow, with a critical value for the transition regime that depends on the geometry.

As an indication, Reynolds' experiments concerning the flow along a straight and smooth pipe showed that the transition regime in this case occurs when Re is between 2000 and 10^4 (see chapter 1.3 in Ref. 3).

Momentum Boundary Layer The momentum boundary layer thickness can be evaluated, using the Reynolds number, by (5.36 in Ref. 2)

$$\delta_M \approx \frac{L}{\sqrt{\mathrm{Re}}} \tag{2}$$

Application in this Tutorial The typical length L of the model is equal to 10 cm so the Reynolds number is evaluated as

$$\mathrm{Re} \approx 10^5 U$$

where U is still unknown. Estimates of this typical velocity are provided later.

Grashof Number

Definition The Grashof number gives the ratio of buoyant to viscous forces. It is defined by

$$\mathrm{Gr} = \frac{\rho^2 g \alpha_p}{\mu^2} \Delta T L^3$$

where g is the gravity acceleration (equal to $9.81\ \mathrm{m/s^2}$) and ΔT is the typical temperature difference.

At atmospheric pressure and at 283 K, air and water have the following properties (Ref. 2).

TABLE 3: THERMOPHYSICAL PROPERTIES OF AIR AND WATER AT 283 K AND ATMOSPHERIC PRESSURE.

Parameter	Description	Air	Water
ρ	Density	$1.25\ \mathrm{kg/m^3}$	$1000\ \mathrm{kg/m^3}$
μ	Dynamic viscosity	$1.76 \cdot 10^{-5}\ \mathrm{N.s.m^{-2}}$	$1.3 \cdot 10^{-3}\ \mathrm{N \cdot s/m^2}$
α_p	Coefficient of thermal expansion	$3.55 \cdot 10^{-3}\ \mathrm{K^{-1}}$	$9 \cdot 10^{-5}\ \mathrm{K^{-1}}$

The value of α_p for air was here obtained by the ideal gas approximation:

$$\alpha_p = \frac{1}{T}$$

You can then evaluate the Grashof number by:

$$\mathrm{Gr}_{\mathrm{air}} \approx 10^8 \Delta T L^3 \qquad \mathrm{Gr}_{\mathrm{water}} \approx 10^9 \Delta T L^3$$

In these relations, ΔT is given in kelvins and L in meters.

The Grashof number can also be expressed as the velocity ratio

$$\mathrm{Gr} = \frac{U_0^2}{(\mu/(\rho L))^2}$$

where U_0 is defined by

$$U_0 = \sqrt{g \alpha_p \Delta T L} \tag{3}$$

This quantity can be considered as the typical velocity due to buoyancy forces.

Flow Regime When buoyancy forces are large compared to viscous forces, the regime is turbulent; otherwise it is laminar. The transition between these two regimes is indicated by the critical order of the Grashof number which is 10^9 (see Figure 7.7 in Ref. 2).

Application in this Tutorial In this tutorial, ΔT is equal to 10 K so the Grashof number is about 10^7 which indicates that a laminar regime is expected.

Table 4 provides the values of the quantities necessary to calculate U_0. This velocity is here of order 10 mm/s.

TABLE 4: THERMOPHYSICAL PROPERTIES OF WATER AT 283 K USED IN THE GRASHOF NUMBER.

PARAMETER	DESCRIPTION	VALUE
g	Gravitational acceleration	9.81 m/s^2
α_p	Coefficient of thermal expansion	$9\cdot10^{-5}$ K^{-1}

Rayleigh Number

Definition The Rayleigh number is another indicator of the regime. It is defined by

$$\mathrm{Ra} = \frac{\rho^2 g \alpha_p C_p}{\mu k}\Delta T L^3$$

so it is similar to the Grashof number except that it accounts for the thermal diffusivity, k, given by

$$\kappa = \frac{k}{\rho C_p}$$

Note: The Rayleigh number can be expressed in terms of the Prandtl and the Grashof numbers through the relation $\mathrm{Ra} = \mathrm{Pr}\,\mathrm{Gr}$.

At atmospheric pressure and at 300 K, you can use the approximations of Ra below for air and water (A.4 in Ref. 1)

$$\mathrm{Ra}_{\mathrm{air}} \approx 10^8 \Delta T L^3 \qquad \mathrm{Ra}_{\mathrm{water}} \approx 10^{10} \Delta T L^3$$

In these relations, ΔT is given in kelvins and L in meters.

Using Equation 3, the Rayleigh number can be rewritten as the velocity ratio

$$\mathrm{Ra} = \frac{U_0^2}{(\nu/L)(\kappa/L)}$$

where the ratio κ/L can be seen as a typical velocity due to thermal diffusion.

Flow Regime Like the Grashof number, a critical Rayleigh value indicates the transition between laminar and turbulent flow. For vertical plates, this limit is about 10^9 (9.23 in Ref. 1).

Typical Velocity Because the viscous forces limit the effects of buoyancy, U_0 may give an overestimated typical velocity. Another approach (see 7.25 in Ref. 2) is to use U_1 instead, defined by

$$U_1 = \frac{\kappa}{L}\sqrt{\mathrm{Ra}}$$

that is

$$U_1 = \frac{k}{\rho C_p L}\sqrt{\mathrm{Ra}} \qquad (4)$$

or

$$U_1 = \frac{U_0}{\sqrt{\mathrm{Pr}}}$$

This should be a more accurate estimate of U because the fluid's thermal diffusivity and viscosity are used in the calculations. From now on, U_1 is the preferred estimate of U.

Thermal Boundary Layer The Rayleigh number can be used to estimate the thermal boundary layer thickness, δ_T. When Pr is of order 1 or greater, it is approximated by the formula (7.25c in Ref. 2)

$$\delta_T \approx \frac{L}{\sqrt[4]{\mathrm{Ra}}} \qquad (5)$$

Application in this Tutorial Here, Ra is of order 10^8. The laminar regime is confirmed but the Rayleigh number found is near the transition zone. The thermal boundary layer thickness is then found to be of order 1 mm and U_1 of order 10 mm/s.

Synthesis

To prepare the simulation, it is very useful to follow the steps below that give indications of what results to expect. It is important to remember that the quantities computed here are only order of magnitude estimates, which should not be considered with more than one significant digit.

First evaluate the Grashof and Rayleigh numbers. If they are significantly below the critical order of 10^9, the regime is laminar. In this case, Equation 3 or Equation 4 provide estimates of the typical velocity U that you can use to validate the model after performing the simulation.

According to Equation 1, the Prandtl number determines the relative thickness of the thermal boundary layer and the outer layer. Equation 2 and Equation 5 then provide orders of magnitude of the thicknesses. When defining the mesh, refinements have to be done at the boundary layers by, for instance, inserting three to five elements across the estimated thicknesses.

Here, Gr and Ra are 10^7 and 10^8, respectively, and thus below the critical value of 10^9 for vertical plates. A laminar regime is therefore expected but because these values are not significantly below 10^9, convergence is not straightforward. In this regime, the estimates U_0 and U_1 of the typical velocity are both of the order 10 mm/s.

For water at 283 K, Pr is about 10 so δ and δ_{T} are of same orders of magnitude. Here, δ_{T} is of the order 1 mm.

The Reynolds number calculated with U_1 is about 10^3, which confirms that the model is close to the transition regime. Using U_1 and Equation 2, the momentum boundary layer thickness δ_{M} is found to be about 1 mm.

2D MODEL

Figure 3 shows the velocity distribution in the square cavity.

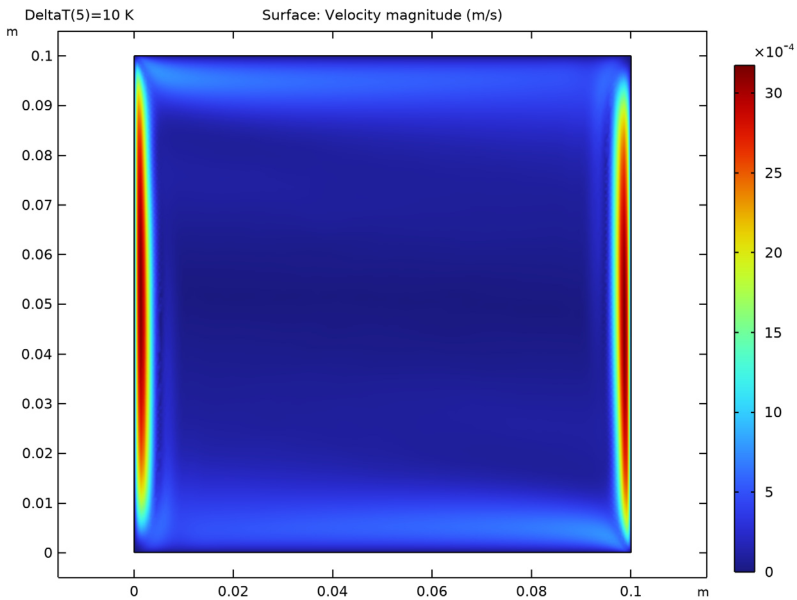


Figure 3: Velocity magnitude for the 2D model.

The regions with faster velocities are located at the lateral boundaries. The maximum velocity is 4 mm/s which is in agreement with the estimated typical velocity U_1 of order 10 mm/s. According to Figure 4, the momentum boundary layer thickness is of order 1 mm, as calculated previously.

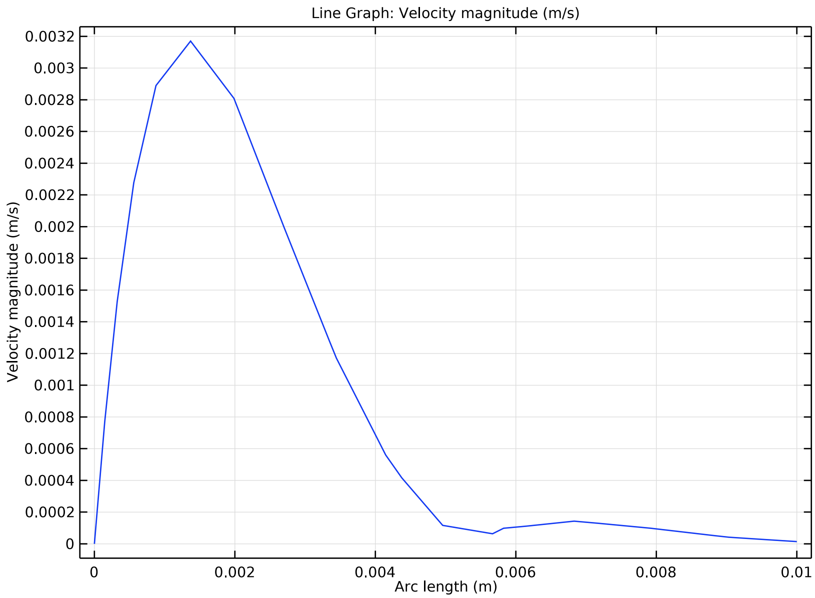


Figure 4: Velocity profile at the left boundary.

Figure 5 shows the temperature field (surface) and velocity field (arrows) of the 2D model.

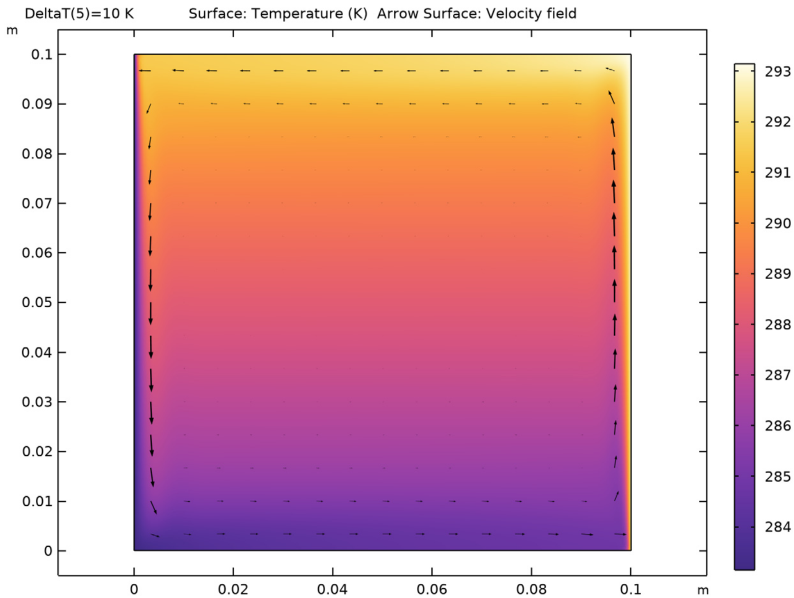


Figure 5: Temperature field (surface plot) and velocity (arrows) for the 2D model.

A large convective cell occupies the whole square. The fluid flow follows the boundaries. As seen in Figure 3, it is faster at the vertical plates where the highest variations of temperature are located. The thermal boundary layer is of the order 1 mm according to

Figure 6, which is in agreement with the estimate provided by Equation 5. The outer layer is slightly thicker than the boundary layer.

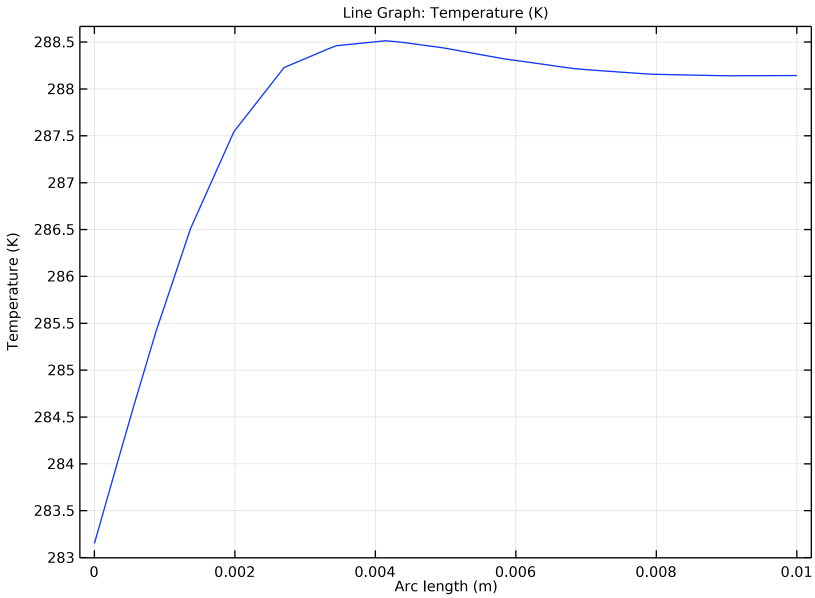


Figure 6: Temperature profile at the left boundary.

3D MODEL

Figure 7 illustrates the velocity plot parallel to the heated plates.

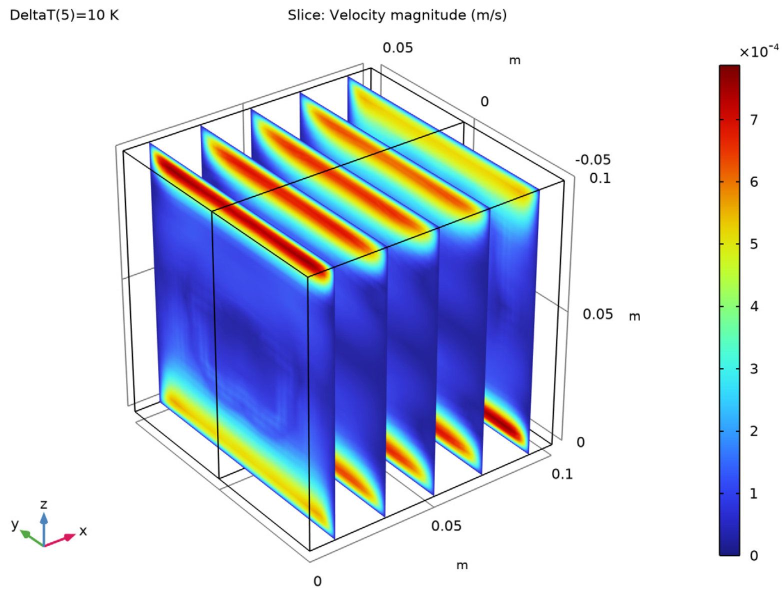


Figure 7: Velocity magnitude field for the 3D model, slices parallel to the heated plates.

A second velocity magnitude field is shown in Figure 8. The plot is close to what was obtained in 2D in Figure 3.

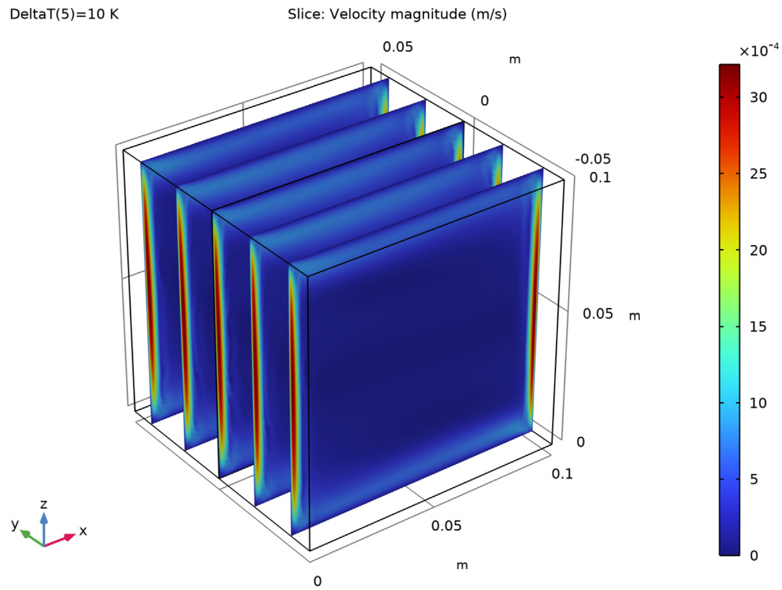


Figure 8: Velocity magnitude field for the 3D model, slices perpendicular to the heated plates.

In Figure 9, velocity arrows are plotted on temperature surface at the middle vertical plane parallel to the plates.

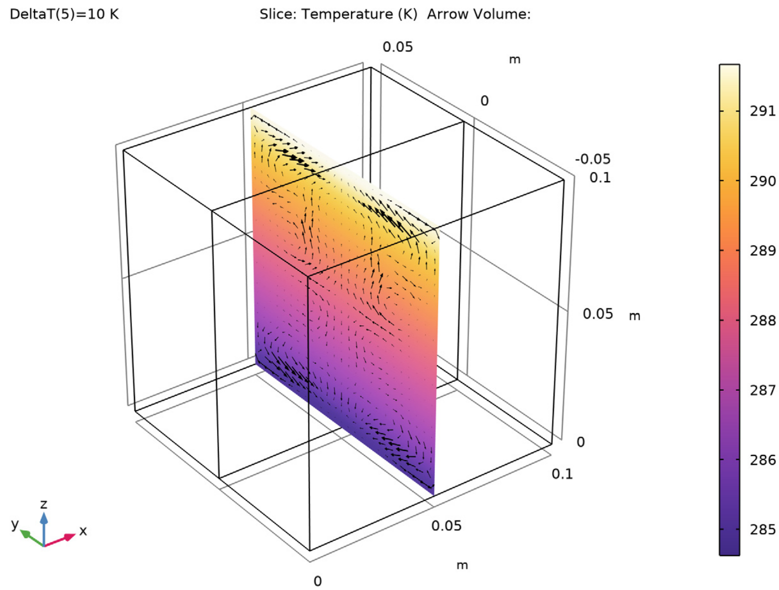


Figure 9: Temperature (surface plot) and velocity (arrows) fields in the cubic cavity, for a temperature difference of 10 K between the vertical plates.

New small convective cells appear on the vertical planes parallel to the plates at the four corners. They are more visible at lower Gr values, that is, far from the transition regime. In Figure 10, the temperature difference between the vertical plates is reduced to 1 K and 0.1 K to decrease the Grashof number to 10^5 and 10^6.

Observe the bigger cells at the four corners of the plane.

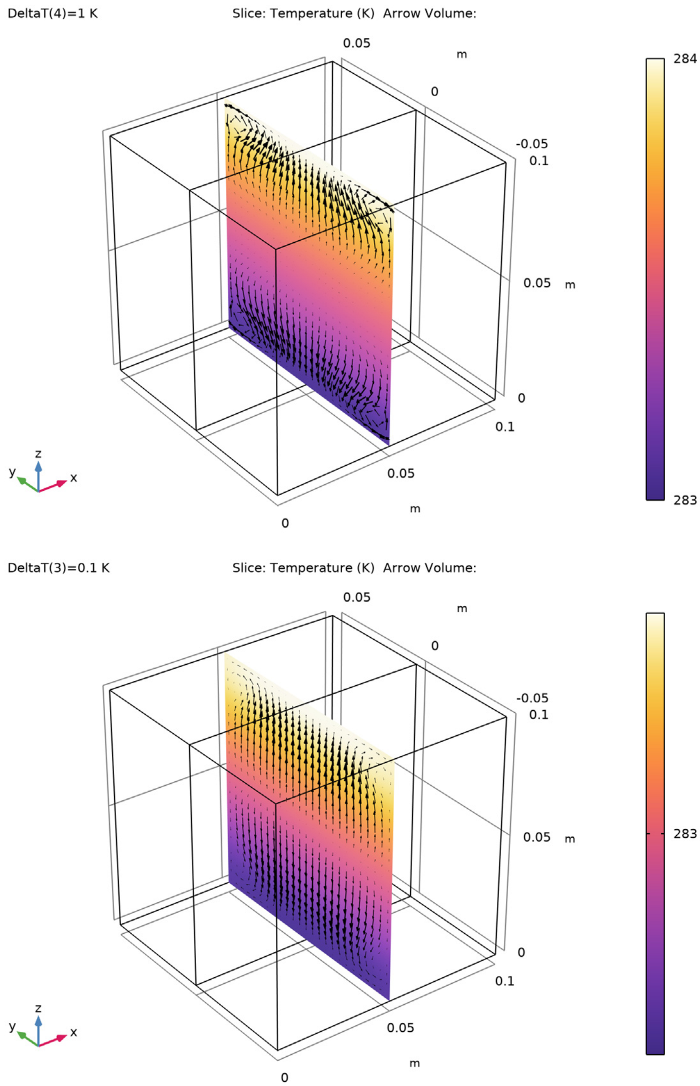


Figure 10: Temperature (surface plot) and velocity (arrows) fields in the cubic cavity, with, for a temperature difference of 1 K (top) and 0.1 K (bottom) between the vertical plates.

Notes About the COMSOL Implementation

The material properties for water are available in the Material Library. The density and dynamic viscosity are functions of the temperature.

At high Gr values, using a good initial condition becomes important in order to achieve convergence. Moreover, a well-tuned mesh is needed to capture the solution, especially the temperature and velocity changes near the walls. Use the Stationary study step's continuation option with ΔT as the continuation parameter to get a solver sequence that uses previous solutions to estimate the initial condition. For this tutorial, it is appropriate to ramp up ΔT from 10^{-3} K to 10 K, which corresponds to a Grashof number range of 10^3–10^7. At $\mathrm{Gr} = 10^3$, the model is easy to solve. The regime is dominated by conduction and viscous effects. At $\mathrm{Gr} = 10^7$, the model becomes more difficult to solve. The regime is greatly influenced by convection and buoyancy.

To get a well-tuned mesh when Gr reaches 10^7, the element size near the prescribed temperature boundaries has to be smaller than the momentum and thermal boundary layer thicknesses, which are of order 1 mm according to Equation 2 and Equation 5. It is recommended to have three to five elements across the layers when using P1 elements (the default setting for fluid flows).

References

1. F.P. Incropera, D.P. DeWitt, T.L. Bergman, and A.S. Lavine, *Fundamentals of Heat and Mass Transfer*, 6th ed., John Wiley & Sons, 2006.

2. A. Bejan, *Heat Transfer*, John Wiley & Sons, 1985.

3. P.A. Davidson, *Turbulence: An Introduction for Scientists and Engineers*, Oxford University Press, 2004.

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

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>Nonisothermal Flow>Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

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

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
L	10[cm]	0.1 m	Square side length
DeltaT	10[K]	10 K	Temperature difference
Tc	283.15[K]	283.15 K	Low temperature
Th	Tc+DeltaT	293.15 K	High temperature
rho	1000[kg/m^3]	1000 kg/m^3	Density
mu	1.3e-3[N*s/m^2]	0.0013 N·s/(m·m)	Dynamic viscosity
k	0.58[W/(m*K)]	0.58 W/(m·K)	Thermal conductivity
Cp	4.2[kJ/(kg*K)]	4200 J/(kg·K)	Heat capacity
alpha	9e-5[1/K]	9E-5 1/K	Coefficient of thermal expansion
U0	sqrt(g_const* alpha*DeltaT*L)	0.029709 m/s	Typical velocity due to buoyancy
U1	U0/sqrt(Pr)	0.0096828 m/s	Typical velocity estimation
Pr	mu*Cp/k	9.4138	Prandtl number
Gr	(U0*rho*L/mu)^2	5.2225E6	Grashof number
Ra	Pr*Gr	4.9163E7	Rayleigh number
Re0	rho*U0*L/mu	2285.3	Reynolds number approximation with U0
Re1	rho*U1*L/mu	744.83	Reynolds number approximation with U1
eps_t	L/(Ra)^0.25	0.0011942 m	Thermal boundary layer thickness
eps_m	L/sqrt(Re1)	0.0036641 m	Momentum boundary layer thickness

The Grashof and Rayleigh numbers should be less than 10^9, indicating that a laminar regime is expected.

GEOMETRY 1

Square 1 (sq1)

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

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

3 In the **Side length** text field, type `L`.

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>Water, liquid**.

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

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

LAMINAR FLOW (SPF)

In order to ensure mass conservation, as the volume is constant, the water density cannot depend only on the temperature. It has to be either constant or pressure and temperature dependent. Select the **Incompressible flow** option to define a constant density evaluated from the material properties at the reference pressure and temperature.

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

2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Incompressible flow**.

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

Pressure Point Constraint 1

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

2 Select Point 2 only.

Fixing the pressure at an arbitrary point is necessary to define a well-posed model.

HEAT TRANSFER IN FLUIDS (HT)

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

2 In the **Settings** window for **Heat Transfer in Fluids**, locate the **Physical Model** section.

3 In the T_{ref} text field, type `(Tc+Th)/2`.

Define the initial temperature as the mean value between the high and low temperature values.

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 `(Tc+Th)/2`.

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 `Tc`.

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 `Th`.

Now, modify the default mesh size settings to ensure that the mesh satisfies the criterion discussed in the Introduction section.

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.

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**.

4 Click **Build All**.

STUDY 1

Because the Grashof number is near the critical value of around 10^9, the model is highly nonlinear. To achieve convergence, use continuation to ramp up the temperature difference value from 10^{-3} K to 10 K, which corresponds to a Grashof number from 10^3 to 10^7.

Step 1: Stationary

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	Parameter unit
DeltaT (Temperature difference)	1e-3 1e-2 1e-1 1 10	K

6 Click the **Show More Options** button in the **Model Builder** toolbar.

7 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

8 Click **OK**.

The pseudo time stepping option is generally useful to help the convergence of a stationary flow model. However, a continuation approach is already used here. In this precise model, disabling the pseudo time stepping option improves the convergence. Follow the instructions below to do so.

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**, click to expand the **Advanced Settings** section.

3 Find the **Pseudo time stepping** subsection. From the **Use pseudo time stepping for stationary equation form** list, choose **Off**.

STUDY 1

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

The first default plot group shows the velocity magnitude as in Figure 3. Notice the high velocities near the lateral walls due to buoyancy effects.

RESULTS

Temperature (ht)

The third default plot shows the temperature distribution. Add an arrow plot of the velocity field to see the correlations between velocity and temperature, as in Figure 5.

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

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

Arrow Surface 1

1 In the **Temperature (ht)** toolbar, click **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.

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

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

In the following steps, the temperature and velocity profiles are plotted near the left boundary in order to estimate the boundary layer thicknesses of the solution.

Cut Line 2D 1

1 In the **Results** toolbar, click **Cut Line 2D**.

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

3 In row **Point 1**, set **y** to `5[cm]`.

4 In row **Point 2**, set **x** to `1[cm]`, and **y** to `5[cm]`.

5 Click **Plot**.

Temperature at Boundary Layer

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

2 In the **Settings** window for **1D Plot Group**, type `Temperature at Boundary Layer` in the **Label** text field.

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

4 From the **Parameter selection (DeltaT)** list, choose **Last**.

Line Graph 1

1 In the **Temperature at Boundary Layer** toolbar, click **Line Graph**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Fluids>Temperature>T - Temperature - K**.

3 In the **Temperature at Boundary Layer** toolbar, click **Plot**.

The thermal boundary layer is around 3 mm.

Velocity at Boundary Layer

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

2 In the **Settings** window for **1D Plot Group**, type `Velocity at Boundary Layer` in the **Label** text field.

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

4 From the **Parameter selection (DeltaT)** list, choose **Last**.

Line Graph 1

1 In the **Velocity at Boundary Layer** toolbar, click **Line Graph**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Laminar Flow>Velocity and pressure>spf.U - Velocity magnitude - m/s**.

3 In the **Velocity at Boundary Layer** toolbar, click **Plot**.

The momentum boundary layer is around 1 mm and the outer layer between 5 mm and 10 mm.

Now create the 3D version of the model.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Fluid Flow>Nonisothermal Flow>Laminar Flow**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

5 Click **Add to Component 2** in the window toolbar.

6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Laminar Flow (spf)** and **Heat Transfer in Fluids (ht)**.

5 Find the **Multiphysics couplings in study** subsection. In the table, clear the **Solve** check box for **Nonisothermal Flow 1 (nitf1)**.

6 Click **Add Study** in the window toolbar.

7 In the **Model Builder** window, click the root node.

8 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the **Width** text field, type `L`.
4 In the **Depth** text field, type `L/2`.
5 In the **Height** text field, type `L`.
6 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>Water, liquid**.
4 Click **Add to Component** in the window toolbar.
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

LAMINAR FLOW 2 (SPF2)

In order to ensure mass conservation, as the volume is constant, the water density cannot depend only on the temperature. It has to be either constant or pressure and temperature dependent. Select the **Incompressible flow** option to define a constant density evaluated from the material properties at the reference pressure and temperature.

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Laminar Flow 2 (spf2)**.
2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.
3 From the **Compressibility** list, choose **Incompressible flow**.
4 Select the **Include gravity** check box.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.
2 Select Point 4 only.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2 Select Boundary 2 only.

HEAT TRANSFER IN FLUIDS 2 (HT2)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Heat Transfer in Fluids 2 (ht2)**.
2 In the **Settings** window for **Heat Transfer in Fluids**, locate the **Physical Model** section.
3 In the T_{ref} text field, type `(Tc+Th)/2`.

Initial Values 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Heat Transfer in Fluids 2 (ht2)** click **Initial Values 1**.
2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3 In the $T2$ text field, type `(Tc+Th)/2`.

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 `Tc`.

Temperature 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2 Select Boundary 6 only.
3 In the **Settings** window for **Temperature**, locate the **Temperature** section.
4 In the T_0 text field, type `Th`.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2 Select Boundary 2 only.

LAMINAR FLOW 2 (SPF2)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Laminar Flow 2 (spf2)**.
2 In the **Settings** window for **Laminar Flow**, click to expand the **Advanced Settings** section.
3 Find the **Pseudo time stepping** subsection. From the **Use pseudo time stepping for stationary equation form** list, choose **Off**.

MULTIPHYSICS

Nonisothermal Flow 2 (nitf2)

1 In the **Model Builder** window, under **Component 2 (comp2)>Multiphysics** click **Nonisothermal Flow 2 (nitf2)**.
2 In the **Settings** window for **Nonisothermal Flow**, locate the **Material Properties** section.
3 Select the **Boussinesq approximation** check box.

MESH 2

To obtain reliable results within in a reasonable computing time, create a structured mesh by following the steps below.

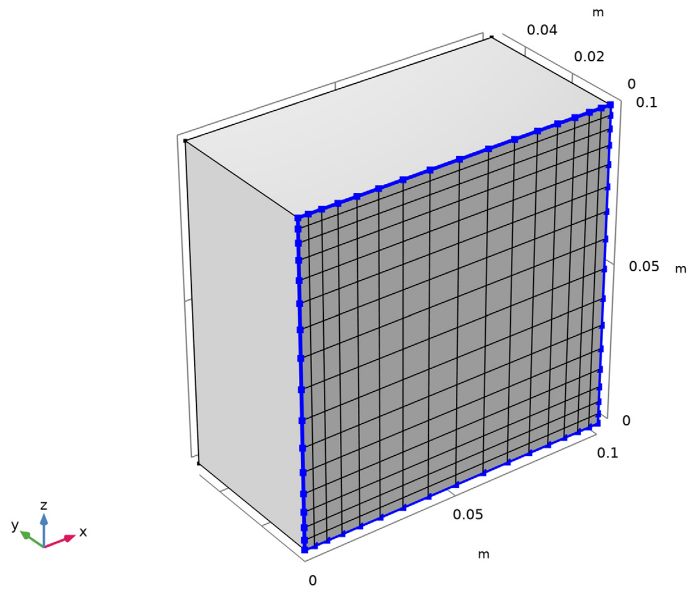
Mapped 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.
2 Select Boundary 2 only.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Edges 1, 3, 5, and 9 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 From the **Distribution type** list, choose **Predefined**.
5 In the **Number of elements** text field, type 16.
6 In the **Element ratio** text field, type 3.
7 Select the **Symmetric distribution** check box.

8 Click **Build Selected**.



The front face mesh has smaller elements near the edges because large variations in velocity and temperature are expected there.

Now extend the front mesh to the remaining structure.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Distribution 1

1. Right-click **Swept 1** and choose **Distribution**.
2. In the **Settings** window for **Distribution**, locate the **Distribution** section.
3. From the **Distribution type** list, choose **Predefined**.
4. In the **Number of elements** text field, type `8`.
5. In the **Element ratio** text field, type `3`.
6. Select the **Reverse direction** check box.

To resolve the boundary layers, use a **Boundary Layers** feature to generate smaller mesh elements near the walls. The thermal boundary layer for the temperature difference of 10 K is approximately 1 mm (see the parameter `eps_t` defined previously). Use this value to define the thickness of the boundary layers.

Boundary Layers 1

In the **Mesh** toolbar, click **Boundary Layers**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 Select Boundaries 1 and 3–6 only.

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

4 In the **Number of layers** text field, type `5`.

5 From the **Thickness specification** list, choose **First layer**.

6 In the **Thickness** text field, type `1[mm]/5`.

7 Click **Build All**.

Finer mesh is needed for accurate results with higher Rayleigh numbers.

MESH 3

In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

Reference 1

1 In the **Mesh** toolbar, click **Modify** and choose **Reference**.

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

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

Refine 1

1 In the **Mesh** toolbar, click **Modify** and choose **Refine**.

2 In the **Settings** window for **Refine**, locate the **Refine Options** section.

3 From the **Refinement method** list, choose **Regular refinement**.

4 Click **Build All**.

First step solves for the smallest Rayleigh numbers using the first mesh.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate 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	Parameter unit
DeltaT (Temperature difference)	1e-3 1e-2 1e-1	K

Step 2: Stationary 1

1 Right-click **Study 2>Step 1: Stationary** and choose **Duplicate**.

Second step solves for the largest Rayleigh numbers using the finest mesh.

2 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.

3 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
DeltaT (Temperature difference)	1 10	K

4 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 2	Mesh 3

5 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

6 From the **Method** list, choose **Solution**.

7 From the **Study** list, choose **Study 2, Stationary**.

8 From the **Selection** list, choose **Last**.

Add a **Combine Solutions** study step that concatenates the two solutions and makes it possible to treat the output as a single parametric study.

Combine Solutions

In the **Study** toolbar, click **Combine Solutions**.

Solution 2 (sol2)

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

The first solution governs the mesh used to discretize the combined solution. The finest mesh is chosen to minimize interpolation error.

Step 3: Combine Solutions

1 In the **Model Builder** window, under **Study 2** click **Step 3: Combine Solutions**.

2 In the **Settings** window for **Combine Solutions**, locate the **Combine Solutions Settings** section.

3 From the **First solution** list, choose **Study 2/Solution Store 2 (sol4)**.

4 From the **Second solution** list, choose **Study 2/Solution Store 1 (sol3)**.

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

2 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node, then click **Fully Coupled 1**.

3 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

4 From the **Nonlinear method** list, choose **Constant (Newton)**.

5 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 2** node, then click **Fully Coupled 1**.

6 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.

7 From the **Nonlinear method** list, choose **Constant (Newton)**.

8 In the **Damping factor** text field, type `0.8`.

While the default solver solves the problem without any issue, GMG is faster for this model.

9 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 2>AMG, nonisothermal flow (nitf2) (merged)** node, then click **Multigrid 1**.

10 In the **Settings** window for **Multigrid**, locate the **General** section.

11 From the **Solver** list, choose **Geometric multigrid**.

12 In the **Study** toolbar, click **= Compute**.

RESULTS

Velocity (spf2)

This default plot group shows the fluid velocity magnitude in only half of the cube. To plot the other half, proceed as follows.

Mirror 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.

3 From the **Plane** list, choose **zx-planes**.

A new dataset containing mirror values is now created. Return to the velocity plot to use this dataset.

Velocity (spf2)

1 In the **Model Builder** window, under **Results** click **Velocity (spf2)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Dataset** list, choose **Mirror 3D 1**.
4 In the **Velocity (spf2)** toolbar, click **Plot**.

Temperature (ht2)

This default plot group shows the temperature distribution. The mirror dataset created previously can be reused here to plot the entire cube.

Velocity, Front Plane

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Velocity, Front Plane` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 1**.

Slice 1

1 In the **Velocity, Front Plane** toolbar, click **Slice**.
2 In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2 (comp2)>Laminar Flow 2> Velocity and pressure>spf2.U - Velocity magnitude - m/s**.
3 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.
4 In the **Velocity, Front Plane** toolbar, click **Plot**.

This slice view shows the velocity magnitude in the same plane as in the 2D model (Figure 8).

Next, plot arrows of the tangential velocity field in the vertical plane parallel to the plates to reproduce Figure 9.

Temperature, 10 K Offset

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Temperature, 10 K Offset` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 1**.

Slice 1

1. In the **Temperature, 10 K Offset** toolbar, click **Slice**.
2. In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2 (comp2)> Heat Transfer in Fluids 2>Temperature>T2 - Temperature - K**.
3. Locate the **Plane Data** section. In the **Planes** text field, type `1`.
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, 10 K Offset** toolbar, click **Plot**.

Temperature, 10 K Offset

In the **Model Builder** window, click **Temperature, 10 K Offset**.

Arrow Volume 1

1. In the **Temperature, 10 K Offset** toolbar, click **Arrow Volume**.
2. In the **Settings** window for **Arrow Volume**, locate the **Expression** section.
3. In the **x-component** text field, type `0`.
4. Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `1`.
5. Find the **y grid points** subsection. In the **Points** text field, type `25`.
6. Find the **z grid points** subsection. In the **Points** text field, type `25`.
7. Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.
8. In the **Temperature, 10 K Offset** toolbar, click **Plot**.

Temperature, 10 K Offset

The arrows follow convective cells at the four corners for a temperature difference of 10 K. Follow the steps below to reproduce Figure 10 and to see these cells when the temperature difference is reduced to 1 K and 0.1 K.

Temperature, 1 K Offset

1. Right-click **Temperature, 10 K Offset** and choose **Duplicate**.
2. In the **Settings** window for **3D Plot Group**, type `Temperature, 1 K Offset` in the **Label** text field.
3. Locate the **Data** section. From the **Parameter value (DeltaT (K))** list, choose **1**.
4. In the **Temperature, 1 K Offset** toolbar, click **Plot**.

Temperature, 0.1 K Offset

1 Right-click **Temperature, 1 K Offset** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Temperature, 0.1 K Offset` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (DeltaT (K))** list, choose **0.1**.

4 In the **Temperature, 0.1 K Offset** toolbar, click **Plot**.