Model created in COMSOL Multiphysics 6.4

Buoyancy Flow in Air

Introduction

This example studies the stationary state of free convection in a cavity filled with air and bounded by two vertical plates. To generate the buoyancy flow, the plates are heated at different temperatures, in a range where the flow remains in a laminar regime.

This model is similar to the model Buoyancy Flow in Water, except that water is replaced by air. The detailed model analysis for the estimation of the flow regime in the Model Analysis section is still applicable here by replacing the water properties by the air properties. Only the values of the indicators used to estimate the flow regime are given in this document, in the Indicators of the Flow Regime section.

The simulation is run in simple 2D and 3D geometries.

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

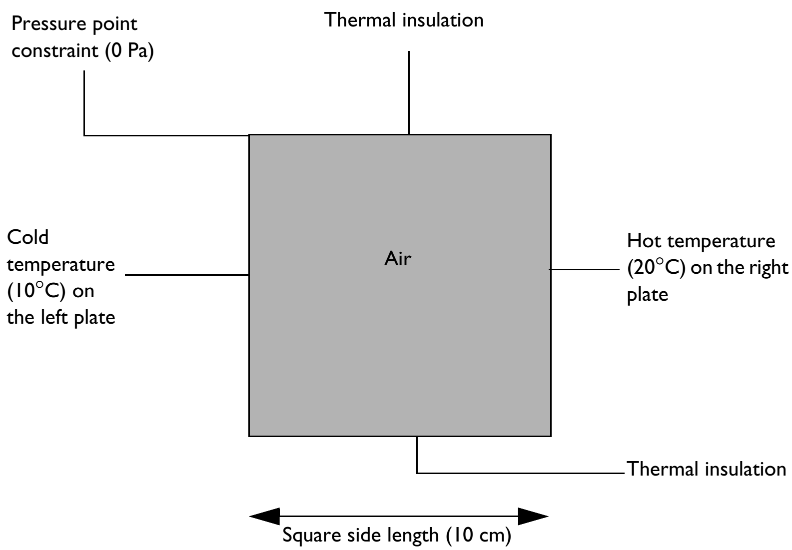

Figure 1: Domain geometry and boundary conditions for the 2D model (square cavity).

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.

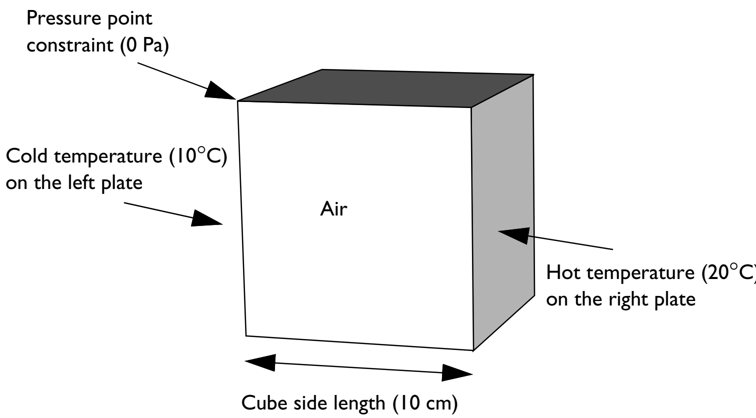

Figure 2: Domain geometry and boundary conditions for the 3D model (cubic cavity).

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 the 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.

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.

INDICATORS OF THE FLOW REGIME

Before starting the simulations, it is recommended to estimate the flow regime. The Grashof number, Rayleigh number, and Prandtl number are then computed and summarized in Table 1 below.

TABLE 1: INDICATORS OF THE FLOW REGIME.

Indicator	Description	Value
Gr	Grashof number	$1.6 \cdot 10^6$
Ra	Rayleigh number	$1.1 \cdot 10^6$
Pr	Prandtl number	0.7

See the model analysis in Buoyancy Flow in Water for more details about the definition of these indicators.

The Grashof and Rayleigh numbers are about 10^6, and thus significantly below the critical value of 10^9 for the flow to become turbulent between vertical plates (see Ref. 1). A laminar regime is therefore expected.

In this regime, the typical velocity due to buoyancy can be estimated by

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

where α_p is the coefficient of thermal expansion and L the typical length. This estimate gives $U_0 = 0.2$ m/s when $\Delta T = 10$ K.

The Prandtl number is lower than 1, which means that the viscous forces should not have any significant limiting effect on buoyancy in this model (see Ref. 2). The estimate of the typical velocity due to buoyancy, taking into account the limiting effect of the viscous forces on buoyancy, gives $U_1 = 0.1$ m/s, which is close to U_0.

Since the Prandtl number is still close to 1, the thermal boundary layer thickness, δ_T, and the shear layer thickness, δ_s, are of the same order of magnitude. They are expressed as (Ref. 2):

$$\delta_\mathrm{T} \approx \frac{L}{\sqrt[4]{\mathrm{Ra} \cdot \mathrm{Pr}}}$$

$$\frac{\delta_\mathrm{s}}{\delta_\mathrm{T}} \approx \sqrt{\mathrm{Pr}}$$

For $\Delta T = 10$ K, these estimates give δ_T and δ_s of about 3 mm, with δ_s slightly smaller than δ_T. You can use these estimates to validate the model after computing the simulation.

The thermophysical properties of air used to compute these indicators are taken from the Material Library. They are listed in Table 2.

TABLE 2: THERMOPHYSICAL PROPERTIES FOR AIR AT 288.15 K.

PARAMETER	DESCRIPTION	VALUE
ρ	Density	1.225 $\mathrm{kg/m^3}$
μ	Dynamic viscosity	$1.789\cdot10^{-5}$ $\mathrm{N\cdot s/m^2}$
k	Thermal conductivity	0.02537 W/(m·K)
C_p	Heat capacity at constant pressure	1.005 kJ/(kg·K)
α_p	Coefficient of thermal expansion	$3.47\cdot10^{-3}$ $\mathrm{K^{-1}}$

These properties are given at 288.15 K which corresponds to the average of the hot and cold plates temperatures. In addition, the coefficient of thermal expansion α_p is computed using the ideal gas approximation:

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

By using the average of the hot and cold plate temperatures in this approximation, you get $\alpha_p = 3.47 \cdot 10^{-3}\ \mathrm{K}^{-1}$.

Results and Discussion

2D MODEL

Figure 3 shows the velocity distribution in the square cavity.

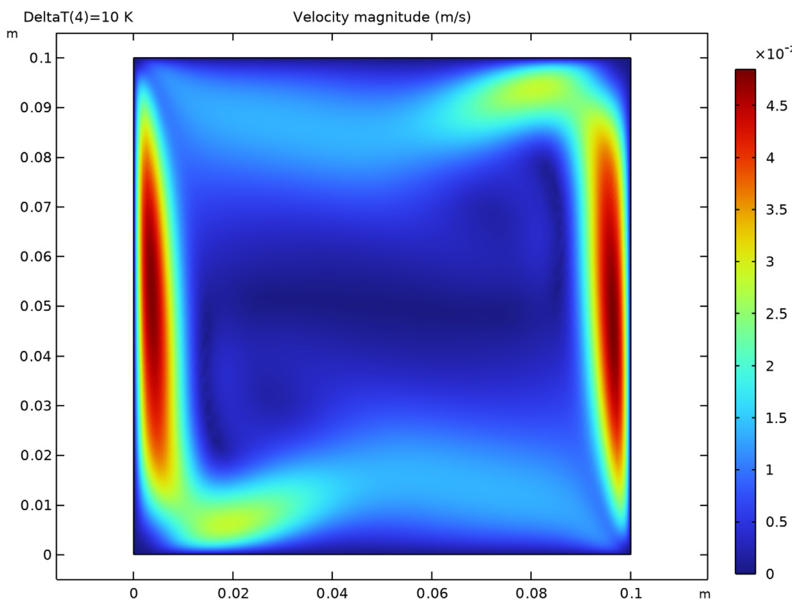

Figure 3: Velocity magnitude for the 2D model.

The regions with faster velocities are located at the lateral boundaries. The maximum velocity is 0.05 m/s which is a bit lower than the estimated typical velocity $U_0 = 0.2$ m/s, but still in the same order of magnitude. According to Figure 4, the shear layer thickness is about 10 mm.

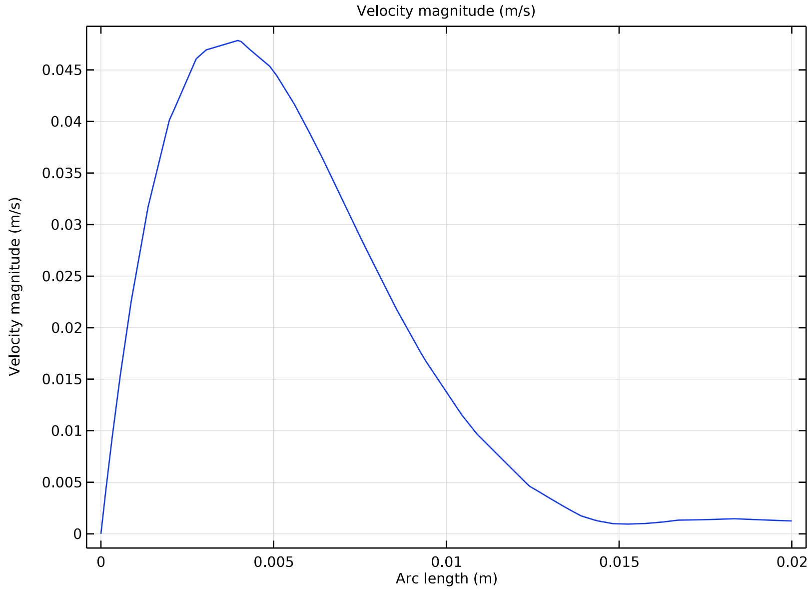

Figure 4: Velocity profile at the left boundary.

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

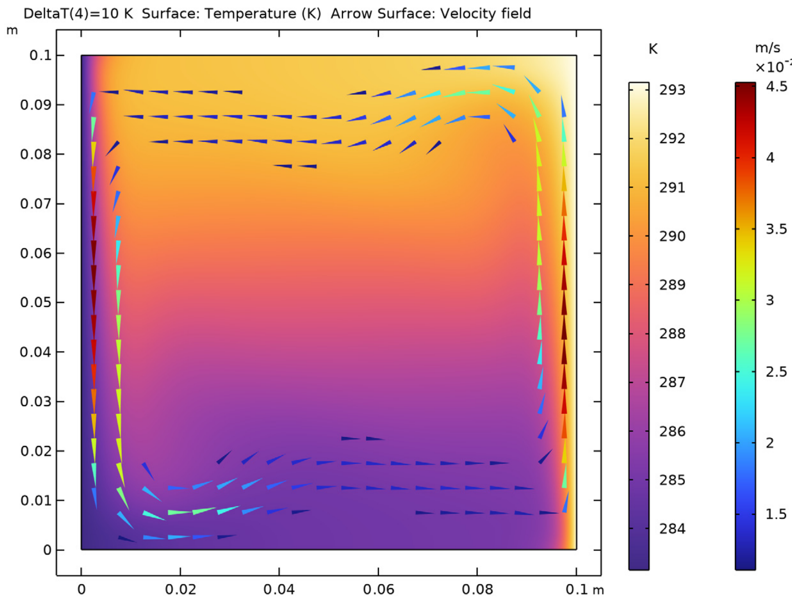

Figure 5: Temperature field (surface plot) and velocity (arrows) for the 2D model.

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 10 mm according to Figure 6, which is in agreement with the order of the estimated value of 3 mm.

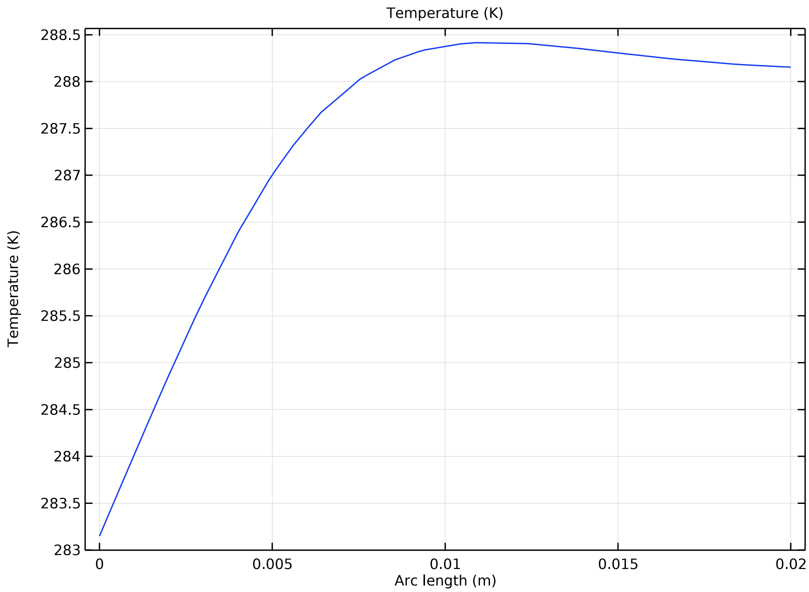

Figure 6: Temperature profile at the left boundary.

3D MODEL

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

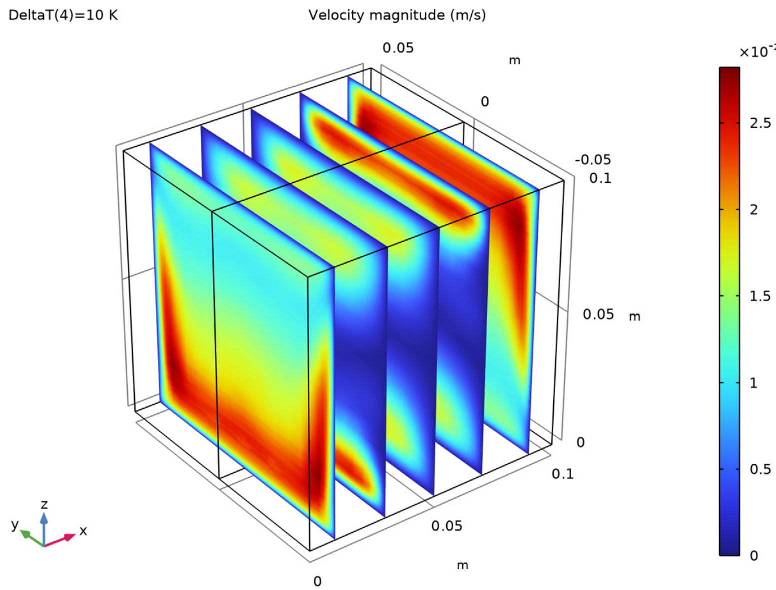

Figure 7: Velocity magnitude field for the 3D model, slices parallel to the heated plates.

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

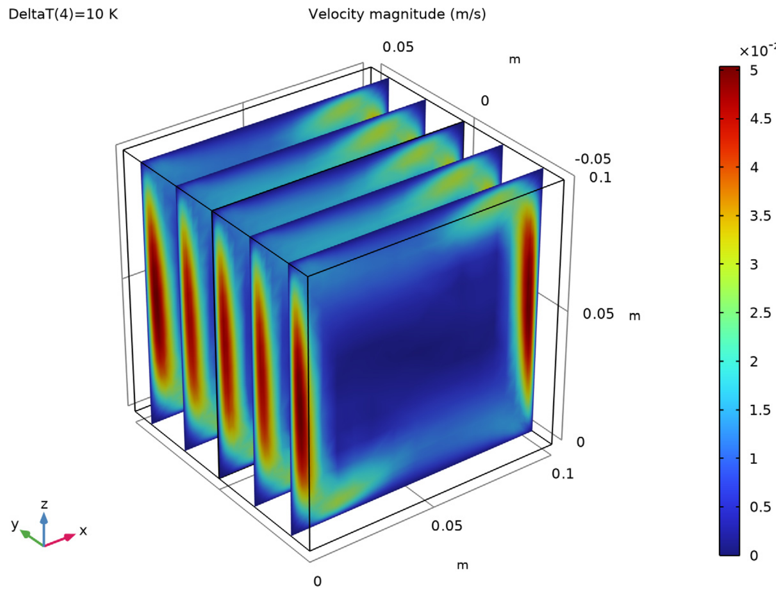

Figure 8: Velocity magnitude field for the 3D model, slices perpendicular to the heated plates.

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

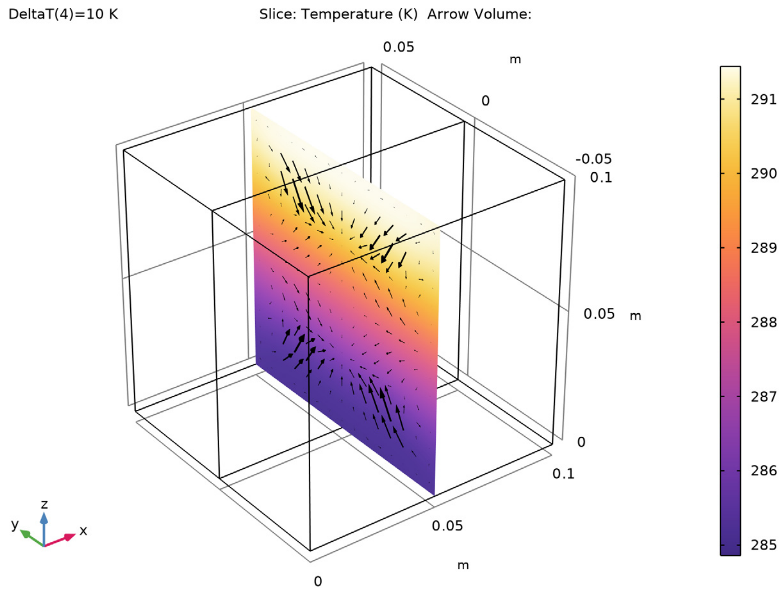

Figure 9: Temperature (surface plot) and velocity (arrows) fields in the cubic cavity, for a temperature difference of 10 K between the vertical plates.

Notes About the COMSOL Implementation

The material properties for air are available in the Material Library.

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^{-2} K to 10 K, which corresponds to a Grashof number range of 10^3–10^6. At $\mathrm{Gr} = 10^3$, the model is easy to solve. The regime is dominated by conduction. At $\mathrm{Gr} = 10^6$, the model becomes more difficult to solve. The regime is more influenced by convection and buoyancy.

To get a well-tuned mesh when Gr reaches 10^6, the element size near the prescribed temperature boundaries has to be smaller than the shear layer and thermal boundary layer thicknesses, which are of order 3 mm. 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, 1993.

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

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
W	L/2	0.05 m	Volume thickness
DeltaT	10[K]	10 K	Temperature difference
Tc	283.15[K]	283.15 K	Low temperature
Th	Tc+DeltaT	293.15 K	High temperature
T_avg	(Tc+Th)/2	288.15 K	Average temperature approximation
p_ref	1[atm]	1.0133E5 Pa	Reference pressure
p0	0[Pa]	0 Pa	Relative pressure constraint
dt_range_start	-2	-2	Range start point of the parametric sweep on DeltaT (log10 of the value in K)
dt_range_stop	1	1	Range end point of the parametric sweep on DeltaT (log10 of the value in K)
rho	1.225[kg/m^3]	1.225 kg/m³	Density at T_avg and 1 atm
mu	1.789e-5[N*s/m^2]	1.789E-5 N·s/(m·m)	Dynamic viscosity at T_avg
k	0.02537[W/(m*K)]	0.02537 W/(m·K)	Thermal conductivity at T_avg
Cp	1.005[kJ/(kg*K)]	1005 J/(kg·K)	Heat capacity at T_avg
alpha	1/T_avg	0.0034704 1/K	Coefficient of thermal expansion at T_avg
U0	sqrt(g_const*alpha*DeltaT*L)	0.18448 m/s	Typical velocity due to buoyancy
U1	U0/sqrt(Pr)	0.21914 m/s	Typical velocity due to buoyancy
Pr	mu*Cp/k	0.70869	Prandtl number

Name	Expression	Value	Description
Gr	(U0*rho*L/mu)^2	1.5957E6	Grashof number
Ra	Pr*Gr	1.1309E6	Rayleigh number
eps_t	L/(Pr*Ra)^0.25	0.0033422 m	Thermal boundary layer thickness
eps_s	eps_t*sqrt(Pr)	0.0028136 m	Shear layer thickness

The Grashof and Rayleigh numbers are 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**.
5 Click the **Zoom Extents** button in the **Graphics** toolbar.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Built-in > Air**.
4 Click the **Add to Component** button in the window toolbar.
5 In the **Materials** 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 air density cannot depend only on the temperature. It has to be either constant or pressure and temperature dependent. In this model the density variations are small and you can make the incompressible assumption.

1 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.
2 From the **Compressibility** list, choose **Incompressible flow**.

3 Select the **Include gravity** checkbox.

The absolute pressure distribution is $p_A = p_{ref} + p$, p being the variable solved for.

The pressure p is automatically initialized at p_0 +ρ gz.

4 In the p_{ref} text field, type `p_ref`.

Set the reference point for gravity at the top of the square, so that the ρgz term of the pressure is 0 at this point.

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

0	x
L	y

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 `p0`.

Fixing the pressure at an arbitrary point is necessary to define a well-posed model. Use a **Pressure Point Constraint** that locks the pressure at the upper-left corner.

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 `p0`.

HEAT TRANSFER IN FLUIDS (HT)

Set the reference temperature to the mean temperature between heated faces.

This value along with the reference pressure determines the reference density used to initialize the pressure p_{init}.

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 `T_avg`.

Initial Values 1

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

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 `T_avg`.

Now define the temperature constraints on the vertical walls.

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`.

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** checkbox.

MESH 1

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

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

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** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

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

6 Click **Range**.

7 In the **Range** dialog, type `dt_range_start` in the **Start** text field.

8 In the **Step** text field, type `1`.

9 In the **Stop** text field, type `dt_range_stop`.

10 From the **Function to apply to all values** list, choose **exp10(x) – Exponential function (base 10)**.

11 Click **Replace**.

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

13 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
DeltaT (Temperature difference)	`10^{range(dt_range_start,1,dt_range_stop)}`	K

14 In the **Model Builder** window, click **Study 1**.

15 In the **Settings** window for **Study**, type `Study 2D` in the **Label** text field.

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

17 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics > Advanced Physics Options**.

18 Click **OK**.

LAMINAR FLOW (SPF)

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 particular model, disabling the pseudo time-stepping option improves the convergence. Follow the instructions below to do so.

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**.

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

RESULTS

Velocity (spf)

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.

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

ROOT

The fourth default plot shows temperature and the velocity field to see the correlations between velocity and temperature. Apply following changes to this plot to get a plot as in Figure 5.

RESULTS

Fluid Flow

1 In the **Model Builder** window, expand the **Results** > **Temperature and Fluid Flow (nitf1)** node, then click **Fluid Flow**.

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

3 Find the **x grid points** subsection. In the **Points** text field, type 20.

4 Find the **y grid points** subsection. In the **Points** text field, type 20.

5 In the **Temperature and Fluid Flow (nitf1)** 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 2**, set **x** to L/5.

4 In row **Point 1**, set **y** to `L/2`.

5 In row **Point 2**, set **y** to `L/2`.

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 **From list**.

5 In the **Parameter values (DeltaT (K))** list box, select **10**.

Line Graph 1

1 Right-click **Temperature at Boundary Layer** and choose **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 thickness is around 10 mm, which is in agreement with the order of the estimated value.

Velocity at Boundary Layer

1 In the **Results** toolbar, click **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 **From list**.

5 In the **Parameter values (DeltaT (K))** list box, select **10**.

Line Graph 1

1 Right-click **Velocity at Boundary Layer** and choose **Line Graph**.

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

The shear layer thickness is around 10 mm, which is in good agreement with the estimated value.

ROOT

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** checkbox for **Study 2D**.
5. Click the **Add to Component 2** button 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** checkboxes for **Laminar Flow (spf)** and **Heat Transfer in Fluids (ht)**.
5. Find the **Multiphysics couplings in study** subsection. In the table, clear the **Solve** checkbox for **Nonisothermal Flow 1 (nitf1)**.
6. Click the **Add Study** button in the window toolbar.
7. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

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 **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in** > **Air**.

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

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

LAMINAR FLOW 2 (SPF2)

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

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

3 Select the **Include gravity** checkbox.

4 In the p_{ref} text field, type `p_ref`.

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

0	x
0	y
L	z

Initial Values 1

1 In the **Model Builder** window, under **Component 2 (comp2)** > **Laminar Flow 2 (spf2)** 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`.

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.

The solution will be evaluated for the geometry (here, a cube) with a symmetry plane at the selected boundary.

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 `T_avg`.

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 `T_avg`.

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.

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** checkbox.

MESH 2

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

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** 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** checkbox.

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** checkbox.

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 3 mm (see the variable `eps_t` defined previously).

Use this value to define the thickness of the boundary layers, and divide the boundary layer in 5 mesh layers of increasing size.

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 `3[mm]/5`.

7 Click **Build All**.

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** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

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

6 Click **Range**.

7 In the **Range** dialog, type `dt_range_start` in the **Start** text field.

8 In the **Step** text field, type `1`.

9 In the **Stop** text field, type `dt_range_stop`.

10 From the **Function to apply to all values** list, choose **exp10(x) – Exponential function (base 10)**.

11 Click **Replace**.

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

13 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
DeltaT (Temperature difference)	`10^{range(dt_range_start,1,dt_range_stop)}`	K

The continuation solver works best for models with linear dependence on the parameter. A more robust alternative for nonlinear applications is to start from the solution for the previous parameter value.

1 From the **Run continuation for** list, choose **No parameter**.

2 From the **Reuse solution from previous step** list, choose **Yes**.

3 In the **Model Builder** window, click **Study 2**.

4 In the **Settings** window for **Study**, type `Study 3D` in the **Label** text field.

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

RESULTS

Velocity (spf2)

The 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. Go back to the velocity plot to use this dataset.

Multislice 1

1 In the **Model Builder** window, expand the **Results** > **Velocity (spf2)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **x-planes** subsection. In the **Planes** text field, type `5`.

4 Find the **y-planes** subsection. In the **Planes** text field, type `0`.

5 Find the **z-planes** subsection. In the **Planes** text field, type `0`.

6 Click to expand the **Quality** section. From the **Resolution** list, choose **Fine**.

Velocity (spf2)

1 In the **Model Builder** window, 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**.

5 Click **Plot**.

Pressure (spf2)

1 In the **Model Builder** window, click **Pressure (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 **Pressure (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.

1 In the **Model Builder** window, click **Temperature (ht2)**.
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 **Temperature (ht2)** toolbar, click **Plot**.

Temperature and Fluid Flow (nitf2)

1 In the **Model Builder** window, click **Temperature and Fluid Flow (nitf2)**.
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 **Temperature and Fluid Flow (nitf2)** toolbar, click **Plot**.

RESULTS

Transparency 1

1 In the **Model Builder** window, expand the **Results** > **Temperature and Fluid Flow (nitf2)** node.
2 Right-click **Wall Temperature** and choose **Transparency**.
3 In the **Temperature and Fluid Flow (nitf2)** toolbar, click **Plot**.

Fluid Flow

1 In the **Model Builder** window, under **Results** > **Temperature and Fluid Flow (nitf2)** click **Fluid Flow**.
2 In the **Settings** window for **Arrow Volume**, locate the **Arrow Positioning** section.
3 Find the **x grid points** subsection. In the **Points** text field, type `12`.
4 Find the **y grid points** subsection. In the **Points** text field, type `12`.
5 Find the **z grid points** subsection. In the **Points** text field, type `12`.
6 In the **Temperature and Fluid Flow (nitf2)** toolbar, click **Plot**.

Velocity, Front Plane

1 In the **Results** toolbar, click **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 Right-click **Velocity, Front Plane** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

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

4 In the **Velocity, Front Plane** toolbar, click **Plot**.

To create the plot shown in Figure 8, follow these steps.

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

Temperature and Velocity, Slice

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Temperature and Velocity, Slice` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 1**.

Slice 1

1 Right-click **Temperature and Velocity, Slice** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Expression** section.

3 In the **Expression** text field, type `T2`.

4 Locate the **Plane Data** section. From the **Entry method** list, choose **Coordinates**.

5 In the **x-coordinates** text field, type `L/2`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **HeatCameraLight**.

Arrow Volume 1

1 In the **Model Builder** window, right-click **Temperature and Velocity, Slice** and choose **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 `15`.

6 Find the **z grid points** subsection. In the **Points** text field, type `15`.

7 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

8 In the **Temperature and Velocity, Slice** toolbar, click **Plot**.

Finally, you can switch to the main flow representation by adding the graphs as follows. Make sure to consider the arrow scale factor for any interpretation.

Slice 2

1 Right-click **Temperature and Velocity, Slice** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

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

4 From the **Entry method** list, choose **Coordinates**.

5 Locate the **Expression** section. In the **Expression** text field, type `T2`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **HeatCameraLight**.

7 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

8 In the **Temperature and Velocity, Slice** toolbar, click **Plot**.

Arrow Volume 2

1 Right-click **Temperature and Velocity, Slice** and choose **Arrow Volume**.

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

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

4 In the **Temperature and Velocity, Slice** toolbar, click **Plot**.