

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:

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

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 I: INDICATORS OF THE FLOW REGIME.

Indicator	Description	Value
Gr	Grashof number	1.6·10 ⁶
Ra	Rayleigh number	1.1·10 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

$$U_0 = \sqrt{g\alpha_p \Delta TL}$$

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

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):

$$\begin{split} \delta_{T} &\approx \frac{L}{\frac{4}{\sqrt{Ra \cdot Pr}}} \\ & \frac{\delta_{s}}{\delta_{T}} \approx \sqrt{Pr} \end{split}$$

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.

PARAMETER	DESCRIPTION	VALUE
ρ	Density	1.225 kg/m ³
μ	Dynamic viscosity	1.789·10 ⁻⁵ N·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	3.47·10 ⁻³ K ⁻¹

TABLE 2: THERMOPHYSICAL PROPERTIES FOR AIR AT 288.15 K.

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 $\alpha_p = 3.47 \cdot 10^{-3} \text{ K}^{-1}$.

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 Gr = 10^3 , the model is easy to solve. The regime is dominated by conduction. At 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

- I In the Model Wizard window, click **Q** 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 **M** Done.

GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.

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
Тс	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_star t	-2	-2	Range start point of the parametric sweep on DeltaT (log10 of the value in K)
dt_range_stop	1	I	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
Ср	1.005[kJ/(kg*K)]	1005 J/(kg·K)	Heat capacity at T_avg
alpha	1/T_avg	0.0034704 I/K	Coefficient of thermal expansion at T_avg
UO	sqrt(g_const* alpha*DeltaT*L)	0.18448 m/s	Typical velocity due to buoyancy
U1	UO/sqrt(Pr)	0.21914 m/s	Typical velocity due to buoyancy
Pr	mu*Cp/k	0.70869	Prandtl number

3 In the table, enter the following settings:

Name	Expression	Value	Description
Gr	(UO*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 I

Square 1 (sq1)

- I 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

- I In the Home toolbar, click 🙀 Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-in>Air.
- 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 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.

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

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

The pressure *p* is automatically initialized at $p_0 + \rho gz$.

5 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.

6 Specify the \mathbf{r}_{ref} vector as

0	x
L	у

Initial Values 1

- I In the Model Builder window, under Component I (compl)>Laminar Flow (spf) click Initial Values I.
- 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 I

- I 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} .

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

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

Now define the temperature constraints on the vertical walls.

Temperature I

- I 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

- I 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)

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

MESH I

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

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Physics-Controlled Mesh section.
- **3** From the **Element size** list, choose **Extra fine**.
- 4 Click 📗 Build All.

STUDY I

Step 1: Stationary

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

6 Click Range.

- 7 In the Range dialog box, 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 explo(x) Exponential function (base 10).
- II Click Replace.
- 12 In the Settings window for Stationary, locate the Study Extensions section.
- **I3** In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
DeltaT (Temperature difference)	10^{range(dt_range_star t,1,dt_range_stop)}	К

I4 In the Model Builder window, click Study I.

- IS In the Settings window for Study, type Study 2D in the Label text field.
- **I6** Click the **5** Show More Options button in the Model Builder toolbar.
- I7 In the Show More Options dialog box, in the tree, select the check box for the node Physics>Advanced Physics Options.

I8 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.

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

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.

Arrow Surface 1

- I In the Model Builder window, right-click Temperature (ht) and choose 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

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

Temperature at Boundary Layer

- I In the Results toolbar, click \sim ID Plot Group.
- 2 In the Settings window for ID 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 I.
- 4 From the Parameter selection (DeltaT) list, choose From list.
- 5 In the Parameter values (DeltaT (K)) list, select 10.

Line Graph 1

- I Right-click Temperature at Boundary Layer and choose Line Graph.
- 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 I (compl)>
 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

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID 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 I.
- 4 From the Parameter selection (DeltaT) list, choose From list.
- 5 In the Parameter values (DeltaT (K)) list, select 10.

Line Graph 1

- I 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

- I 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 2D.
- 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

- I In the Home toolbar, click $\sim\sim$ 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 I (nitfl)**.
- 6 Click Add Study in the window toolbar.
- 7 In the Home toolbar, click $\stackrel{\text{res}}{\longrightarrow}$ Add Study to close the Add Study window.

GEOMETRY 2

Block I (blkI)

- I 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

- I In the Home toolbar, click 👯 Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select **Built-in>Air**.
- 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)

I 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.
- **5** In the *p*_{ref} text field, type p_ref.
- **6** Specify the \mathbf{r}_{ref} vector as

0	x
0	у
L	z

Initial Values 1

- I In the Model Builder window, under Component 2 (comp2)>Laminar Flow 2 (spf2) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the *p* text field, type p0.

Pressure Point Constraint I

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

Symmetry I

- I 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)

- I 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

- I In the Model Builder window, under Component 2 (comp2)>Heat Transfer in Fluids 2 (ht2) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the T2 text field, type T_avg.

Temperature I

- I 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

- I 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 I

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

MULTIPHYSICS

Nonisothermal Flow 2 (nitf2)

- I 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 a short computing time, create a structured mesh by following the steps below.

Mapped I

- I In the Mesh toolbar, click \bigwedge Boundary and choose Mapped.
- **2** Select Boundary 2 only.

Distribution I

- I Right-click Mapped I 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.

Swept I

In the Mesh toolbar, click 🦓 Swept.

Distribution I

- I Right-click Swept I 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 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 Moundary Layers.

Boundary Layer Properties

- I 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

I In the Model Builder window, under Study 2 click Step I: 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)		К

- 6 Click Range.
- 7 In the Range dialog box, 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 explo(x) Exponential function (base 10).
- II Click Replace.

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

I3 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
DeltaT (Temperature difference)	10^{range(dt_range_star t,1,dt_range_stop)}	К

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.

- I 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 **Home** 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 I

- I 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.

Slice

- I In the Model Builder window, expand the Results>Velocity (spf2) node, then click Slice.
- 2 In the Settings window for Slice, click to expand the Quality section.
- **3** From the **Resolution** list, choose **Fine**.

Velocity (spf2)

- I 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 I.
- 4 In the Velocity (spf2) toolbar, click 💿 Plot.
- 5 Click 💽 Plot.

Pressure (spf2)

- I 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 I.
- **4** In the **Pressure (spf2)** toolbar, click **O 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.

- I 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 I.
- 4 In the Temperature (ht2) toolbar, click **O** Plot.

Isothermal Contours (ht2)

- I In the Model Builder window, click Isothermal Contours (ht2).
- 2 In the Settings window for 3D Plot Group, locate the Data section.

- 3 From the Dataset list, choose Mirror 3D I.
- 4 In the Isothermal Contours (ht2) toolbar, click 💿 Plot.

Velocity, Front Plane

- I 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 I.

Slice 1

- I 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.

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, Velocity

- I In the Home toolbar, click 🚛 Add Plot Group and choose 3D Plot Group.
- 2 In the Settings window for 3D Plot Group, type Temperature, Velocity in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Mirror 3D I.

Slice 1

- I Right-click Temperature, Velocity 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. Click Change Color Table.
- 7 In the Color Table dialog box, select Thermal>HeatCameraLight in the tree.
- 8 Click OK.

Arrow Volume 1

- I In the Model Builder window, right-click Temperature, Velocity 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, Velocity toolbar, click **I** 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

- I Right-click Temperature, Velocity 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. Click Change Color Table.
- 7 In the Color Table dialog box, select Thermal>HeatCameraLight in the tree.
- 8 Click OK.
- 9 In the Settings window for Slice, click to expand the Inherit Style section.
- **IO** From the **Plot** list, choose **Slice I**.
- II In the **Temperature**, **Velocity** toolbar, click **O** Plot.

Arrow Volume 2

- I Right-click Temperature, Velocity 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**, **Velocity** toolbar, click **I** Plot.