Model created in COMSOL Multiphysics 6.4

Marangoni Effect

Introduction

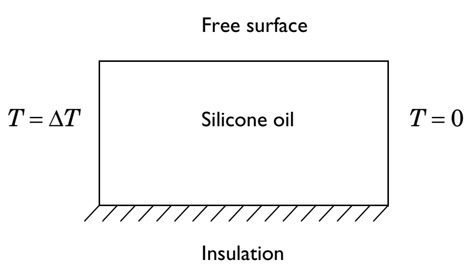
Marangoni convection occurs when the surface tension of an interface (generally liquid-air) depends on the concentration of a species or on the temperature distribution. In the case of temperature dependence, the Marangoni effect is also called thermo-capillary convection. It is of primary importance in the fields of:

- Welding
- Crystal growth
- Electron beam melting of metals

Direct experimental studies are not easy to do in these systems because the materials are often metals and temperatures are very high. One possibility is to replace the real system with an experimental setup using a transparent liquid at ambient temperatures.

Model Definition

This tutorial describes the 2D stationary behavior of a vessel filled with silicone oil, for which the thermo-physical properties are known. The aim of the study is to compute the temperature field that induces a flow through the Marangoni effect. The model shows this effect using the simple geometry in the figure below.



GOVERNING EQUATIONS

A stationary momentum balance equation describes the velocity field and the pressure distribution (Navier–Stokes equations, see the section *Incompressible Flow* in the *COMSOL Multiphysics Reference Manual*). To include the heating of the fluid, the fluid flow is coupled to an energy balance.

You can use the Boussinesq approximation to include the effect of temperature on the velocity field. In this approximation, variations in temperature produce a buoyancy force (or Archimedes' force) that lifts the fluid as described in the sections *Gravity* and *The Boussinesq Approximation* in the *CFD Module User's Guide*.

The following equation describes the forces that the Marangoni effect induces on the interface (liquid/air):

$$\eta\frac{\partial u}{\partial y} = \gamma\frac{\partial T}{\partial x} \qquad (1)$$

Here γ is the temperature derivative of the surface tension (N/(m·K)). Equation 1 states that the shear stress on a surface is proportional to the temperature gradient (Ref. 1).

Notes About the COMSOL Implementation

To solve the momentum and energy balance equations, use the predefined Nonisothermal Flow multiphysics coupling. It automatically couples a Laminar Flow interface for the fluid flow to a Heat Transfer in Fluids interface for the heat transfer by convection and conduction in each direction:

- The Boussinesq approximation means that an expression including temperature acts as a force in the y direction in the momentum balance.
- The convective heat transfer depends on the velocities from the momentum balance.

This means that you must solve the coupled system directly using the nonlinear solver.

To impose the condition that the shear stress is proportional to the temperature gradient on the surface, use the Marangoni Effect multiphysics feature in the Multiphysics node.

Results

The Marangoni effect becomes more pronounced as the temperature difference increases:

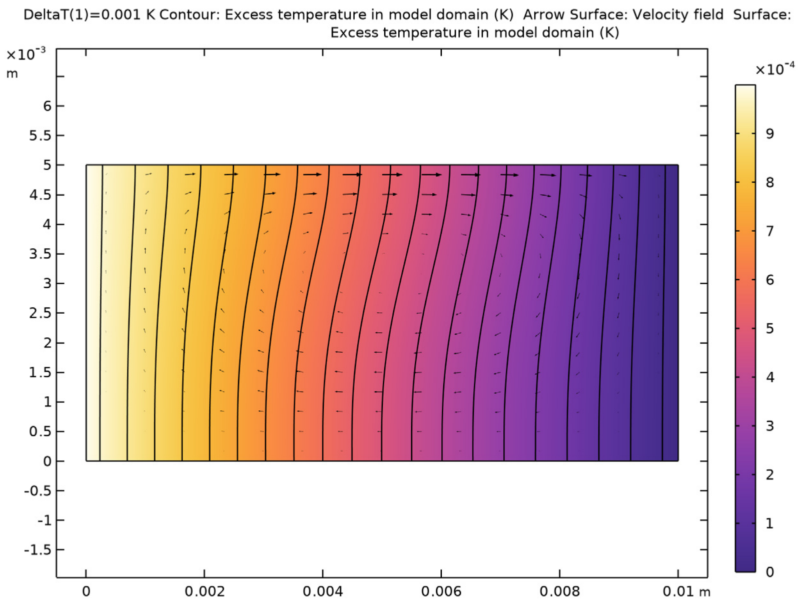


Figure 1: Marangoni convection with a temperature difference of 0.001 K.

For the very low temperature difference of 0.001 K, the temperature field is almost decoupled from the velocity field. Therefore, the temperature decreases almost linearly from left to right.

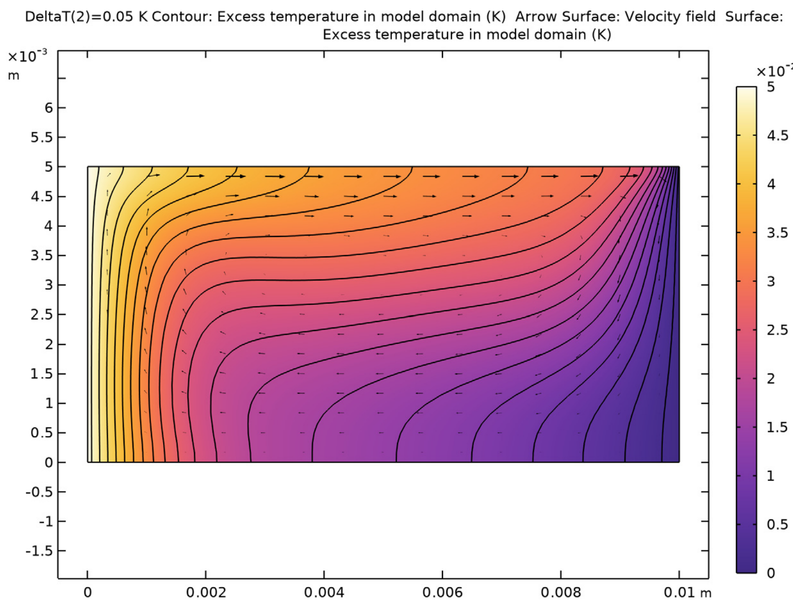


Figure 2: Marangoni convection with a temperature difference of 0.05 K.

For the temperature difference of 0.05 K notice how the Marangoni convection influences the flow of fluid and the distribution of temperature. The temperature is no longer decreasing linearly and you can clearly see the advection of the isotherms caused by the flow.

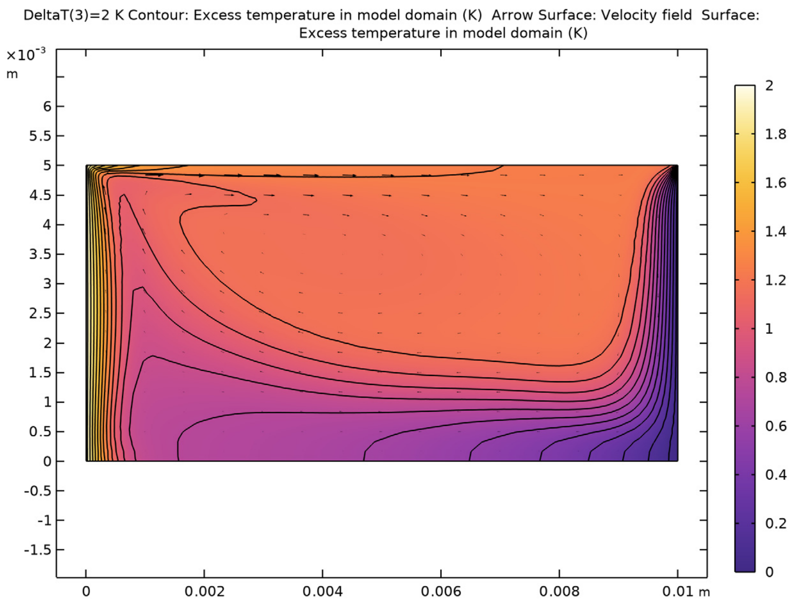


Figure 3: Marangoni convection with a temperature difference of 2 K.

At higher temperature differences (2 K in Figure 3 above), the physical coupling between the temperature and the velocity field is clearly visible. The heat conduction is small compared to the convection, and at the surface the fluid accelerates where the temperature gradient is high.

Reference

1. V.G. Levich, *Physicochemical Hydrodynamics*, Prentice-Hall, N.J., 1962.

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

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**.

GEOMETRY 1

Rectangle 1 (r1)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `10[mm]`.

4 In the **Height** text field, type `5[mm]`.

5 In the **Geometry** toolbar, click **Build All**.

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

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `marangoni_effect_parameters.txt`.

DEFINITIONS

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Unit**	**Description**
`deltaT`	`T-T_right`	K	`Excess temperature in model domain`

This variable is useful when visualizing the model results.

MATERIALS

Silicone Oil

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Silicone Oil` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Dynamic viscosity	mu	`mu1`	Pa·s	Basic
Heat capacity at constant pressure	Cp	`Cp1`	J/(kg·K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`k1`	W/(m·K)	Basic

LAMINAR FLOW (SPF)

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

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

3 Select the **Include gravity** checkbox.

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

Wall 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.

4 From the **Wall condition** list, choose **Slip**.

Pressure Point Constraint 1

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

2 Select Point 1 only.

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 `T_ref`.

Here, `T_ref` is the reference temperature at which the material properties are evaluated. It is defined in **Parameters** under **Global Definitions**.

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 `T_right`.

Temperature 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `T_right`.

Temperature 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `T_right+DeltaT`.

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.

4 From the **Specify density** list, choose **Custom, linearized density**.

5 In the ρ_{ref} text field, type `rho1`.

6 In the $\alpha_{p,0}$ text field, type `alphap1`.

Marangoni Effect 1 (mar1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary** > **Marangoni Effect**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Marangoni Effect**, locate the **Surface Tension** section.

4 In the σ text field, type `gamma*T`.

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 3 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `1e-4`.

Size 2

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Points 2 and 4 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `2e-5`.

Size

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

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

3 From the **Predefined** list, choose **Extra fine**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Maximum element growth rate** text field, type `1.1`.

6 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 (Excess temperature on the left boundary)	`1e-3 5e-2 2`	K

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

RESULTS

To show the temperature field as a surface plot along with overlaid temperature contours and the velocity field using arrows, follow the steps given below.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Heat Transfer in Fluids** > **Isothermal Contours (ht)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Isothermal Contours (ht)

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

2 From the **Parameter value (DeltaT (K))** list, choose **0.001**.

Contour 1

1 In the **Model Builder** window, expand the **Isothermal Contours (ht)** node, then click **Contour 1**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Variables** > **deltaT - Excess temperature in model domain - K**.

3 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

5 Clear the **Color legend** checkbox.

Isothermal Contours (ht)

In the **Model Builder** window, click **Isothermal Contours (ht)**.

Arrow Surface 1

1 In the **Isothermal Contours (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**.

Isothermal Contours (ht)

In the **Model Builder** window, click **Isothermal Contours (ht)**.

Surface 1

1 In the **Isothermal Contours (ht)** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Variables** > **deltaT - Excess temperature in model domain - K**.

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

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

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

Isothermal Contours (ht)

The Marangoni effect becomes more pronounced as the temperature difference increases. Visualize this by changing the **Parameter value** selection.

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

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

3 From the **Parameter value (DeltaT (K))** list, choose **0.05**.

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

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

6 From the **Parameter value (DeltaT (K))** list, choose **2**.

7 In the **Isothermal Contours (ht)** toolbar, click **Plot**.

Follow these steps to visualize the importance of the Marangoni effect on the convection cell.

Convection Cell

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Convection Cell` in the **Label** text field.

3 Click to collapse the **Data** section.

Surface 1

1 In the **Convection Cell** toolbar, click **Surface**.

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

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

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

Convection Cell

In the **Convection Cell** toolbar, click **Streamline**.

Streamline 1

1 In the **Settings** window for **Streamline**, locate the **Expression** section.

2 Select the **Description** checkbox. In the associated text field, type `Velocity field (m/s)`.

3 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Magnitude controlled**.

4 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

Color Expression 1

Right-click **Streamline 1** and choose **Color Expression**.

Convection Cell

1 In the **Settings** window for **2D Plot Group**, click to expand the **Data** section.

2 From the **Parameter value (DeltaT (K))** list, choose **0.001**.

3 In the **Convection Cell** toolbar, click **Plot**.

4 Click **Plot Next**.

5 Click → **Plot Next**.