Free Convection in a Porous Medium

Introduction

This example describes flow in porous media driven by density variations that result from temperature changes. The example comes from Hossain and Wilson (Ref. 1), who use a specialized in-house code to solve this free-convection problem. This COMSOL Multiphysics example reproduces their work using the Brinkman Equations interface and the Heat Transfer in Porous Media interface. The results of this model match those of the published study.

Model Definition

The following figure gives the example geometry. Water in a porous medium layer can move within the layer but not exit from it. Temperatures vary from high to low along the outer edges. Initially the water is stagnant, but temperature gradients alter the fluid density to the degree that buoyant flow occurs. The problem statement specifies that the flow is steady state.

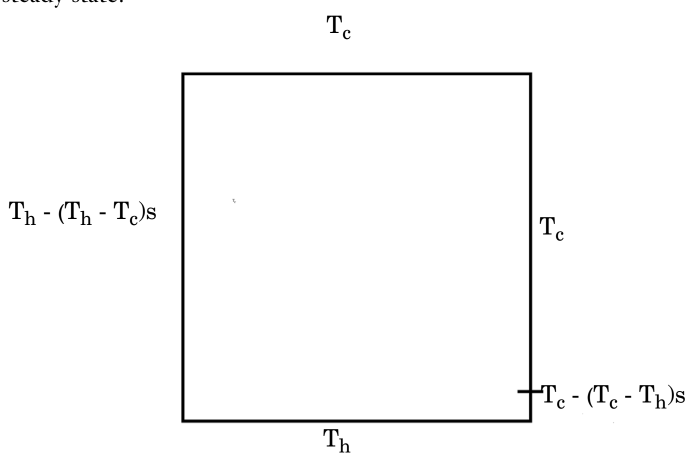

Figure 1: Domain geometry and boundary conditions for the heat balance in the free-convection problem. T_h is a higher temperature than T_c, while s is a variable that represents the relative length of a boundary segment and goes from 0 to 1 along the segment.

Model this free-convection problem by introducing a Boussinesq buoyancy term to Brinkman's momentum equation, and then linking the resulting fluid velocities to the Heat Transfer in Porous Media interface.

The Boussinesq buoyancy term that appears on the right-hand side of the momentum equation accounts for the lifting force due to thermal expansion

$$\frac{\mu}{\kappa}\mathbf{u} + \nabla p - \nabla \cdot \frac{\mu}{\varepsilon}(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) = \rho \mathbf{g} \alpha_p (T - T_c) \tag{1}$$

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

In these expressions, T represents temperature, while T_c is a reference temperature, $\mathbf{g}$ denotes the gravity acceleration, ρ gives the fluid density at the reference temperature, ε is the porosity, and α_p is the fluid's coefficient of volumetric thermal expansion.

The heat balance comes from the heat transfer equation

$$\rho C_{\mathrm{L}} \mathbf{u} \cdot \nabla T - \nabla \cdot (k_{\mathrm{eq}} \nabla T) = 0 \tag{2}$$

where k_{eq} denotes the effective thermal conductivity of the fluid-solid mixture, and C_{L} is the fluid's heat capacity at constant pressure.

The boundary conditions for the Brinkman equations are all no-slip conditions. Using only velocity boundaries gives no information on the pressure within the domain, which means that the example produces estimates of the pressure change instead of the pressure field. However, without any seed information on pressure, the problem is unlikely to converge. The remedy is to arbitrarily fix the pressure at a point in the example using a point constraint. The boundary conditions for the Heat Transfer interface are the series of fixed temperatures shown in Figure 1.

Implementation: Initial Conditions for Boussinesq Approximation

The simple statements in Equation 1 and Equation 2 produce a strong nonlinear problem that represents a difficult convergence task for most nonlinear solvers. To ease the numerical difficulties, let the coefficient of volumetric thermal expansion α_p increase gradually, raising the Rayleigh number of the experiment. When $\alpha_p = 0$, the momentum and temperature equations are uncoupled, so the example converges easily. Then increase α_p, using the previous solution as the initial guess for the next parametric step, and so on, until reaching a Rayleigh number of 10^5. The iteration protocol is an easy process with the parametric solver in COMSOL Multiphysics.

Results

This example reproduces a model reported by Hossain and Wilson (Ref. 1). After extracting the input data from the paper, the author constructed the example in less than an hour, including all the steps from geometry input to postprocessing of the results. Figure 2 shows the temperature distribution throughout the porous slice.

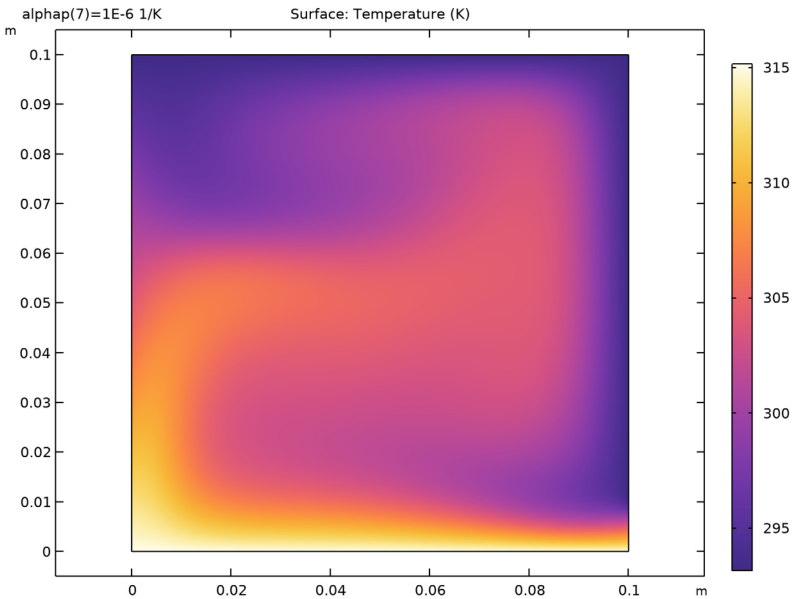

Figure 2: Temperature in a porous structure subjected to temperature gradients and subsequent free convection. The COMSOL Multiphysics simulation is in excellent agreement with published results from Ref. 1.

Figure 3 gives the COMSOL Multiphysics solution for the flow field.

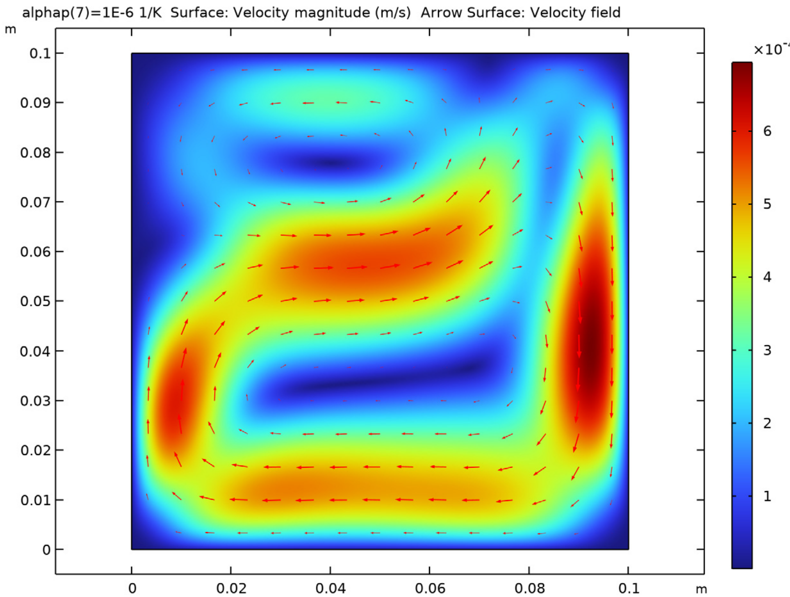

Figure 3: Velocity field for a Rayleigh number of $Ra = 10^5$.

Reference

1. M. Anwar Hossain and M. Wilson, “Natural Convection Flow in a Fluid-saturated Porous Medium Enclosed by Non-isothermal Walls with Heat Generation,” *Int. J. Therm. Sci.*, vol. 41, pp. 447–454, 2002.

Application Library path: `Porous_Media_Flow_Module/Heat_Transfer/convection_porous_medium`

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>Brinkman Equations**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Start by loading parameters from a file. The list contains material parameters and also parameters for setting up the physics.

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 `convection_porous_medium_parameters.txt`.

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`.

Point 1 (pt1)

1 In the **Geometry** toolbar, click **Point**.
2 In the **Settings** window for **Point**, locate the **Point** section.
3 In the **x** text field, type `L`.
4 In the **y** text field, type `L/10`.
5 Click **Build All Objects**.

MATERIALS

With choosing the Nonisothermal Flow, Brinkman Equations multiphysics interface a **Porous Material** is added automatically.

Porous Material 1 (pmat1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials** node, then click **Porous Material 1 (pmat1)**.

2 In the **Settings** window for **Porous Material**, locate the **Homogenized Properties** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	kappa	m^2	Basic

Fluid (pmat1.fluid)

1 In the **Model Builder** window, expand the **Porous Material 1 (pmat1)** node, then click **Fluid (pmat1.fluid)**.

2 In the **Settings** window for **Fluid**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rho0	kg/m^3	Basic
Heat capacity at constant pressure	Cp	Cp0	J/(kg·K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	k0	W/(m·K)	Basic
Dynamic viscosity	mu	mu0	Pa·s	Basic

Solid (pmat1.solid)

1 In the **Model Builder** window, click **Solid (pmat1.solid)**.

2 In the **Settings** window for **Solid**, locate the **Solid Properties** section.

3 In the θ_s text field, type `1-epsilon`.

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

Property	Variable	Value	Unit	Property group
Density	rho	0	kg/m^3	Basic

Property	Variable	Value	Unit	Property group
Heat capacity at constant pressure	Cp	0	J/(kg·K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0	W/(m·K)	Basic

BRINKMAN EQUATIONS (BR)

Continue with setting up the physics. Automatically a weakly compressible formulation of the Brinkman Equations is set up for nonisothermal flow. For the Boussinesq Approximation the incompressible formulation is used, because only density variations in the buoyancy term are considered.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Brinkman Equations (br)**.
2 In the **Settings** window for **Brinkman Equations**, locate the **Physical Model** section.
3 From the **Compressibility** list, choose **Incompressible flow**.
4 Select the **Include gravity** check box.
5 Click to expand the **Advanced Settings** section. Find the **Pseudo time stepping** subsection. From the **Use pseudo time stepping for stationary equation form** list, choose **Off**
 The pseudo time stepping option is generally useful to help the convergence of a stationary flow model. However, a continuation approach will be used here. In this precise model, disabling the pseudo time stepping option improves the convergence.

Since the pressure is not set explicitly by a boundary condition, you need to fix it at least at one point to get a unique solution.

Pressure Point Constraint 1

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

HEAT TRANSFER IN POROUS MEDIA (HT)

Use quadratic elements for the discretization of the temperature field to improve accuracy for this strongly coupled problem.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Porous Media (ht)**.
2 In the **Settings** window for **Heat Transfer in Porous Media**, click to expand the **Discretization** section.
3 From the **Temperature** list, choose **Quadratic Lagrange**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Heat Transfer in Porous Media (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`.

Temperature 1

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

Temperature 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2 Select Boundaries 3 and 5 only.
3 In the **Settings** window for **Temperature**, locate the **Temperature** section.
4 In the T_0 text field, type `Tc`.

Temperature 3

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 `Th-(Th-Tc)*s`.

Temperature 4

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 `Tc-(Tc-Th)*s`.

MULTIPHYSICS

To use the Boussinesq Approximation make the following changes in the **Multiphysics > Nonisothermal Flow1** node.

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.

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

5 In the ρ_{ref} text field, type `rho0`.

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

MESH 1

Use a finer mesh setting to resolve the convection pattern well.

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 **Finer**.

STUDY 1

Step 1: Stationary

Set up an auxiliary continuation sweep for the `alphap` parameter to improve stability.

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 From the **Parameter name** column select `alphap (Fluid volumetric thermal expansion)`.

6 Click **Range**.

7 In the **Range** dialog box, type `-12` in the **Start** text field.

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

9 In the **Stop** text field, type `-6`.

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

11 Click **Replace**.

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

RESULTS

Velocity (br)

The first default plot group shows the velocity magnitude. Refine the resolution for the surface plot to get a smooth velocity field. Add an arrow plot to see the flow direction and compare with Figure 3.

Surface

1. In the **Model Builder** window, expand the **Velocity (br)** node, then click **Surface**.
2. In the **Settings** window for **Surface**, click to expand the **Quality** section.
3. From the **Resolution** list, choose **Finer**.
4. In the **Velocity (br)** toolbar, click **Plot**.

Arrow Surface 1

1. In the **Model Builder** window, right-click **Velocity (br)** and choose **Arrow Surface**.
2. In the **Velocity (br)** toolbar, click **Plot**.

Pressure (br)

Because the cavity is closed, the pressure distribution is solely due to gravity.

Temperature (ht)

The third default plot group shows the temperature field as a surface plot (Figure 2).

Temperature and Fluid Flow (nitf1)

The fourth default plot group shows the velocity field and temperature.