Free Convection in a Porous Medium

Introduction

This example describes subsurface 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.

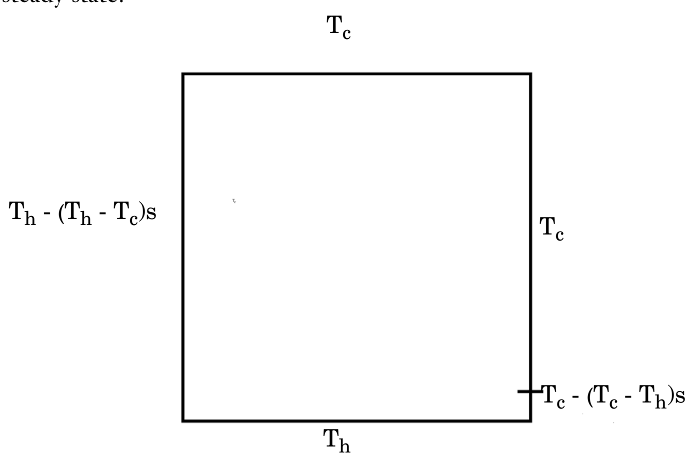

Figure 1: Domain geometry and boundary conditions for the heat balance in the free-convection problem. T_h is a higher temperature than T_c, while s is a variable that represents the relative length of a boundary segment and goes from 0 to 1 along the segment.

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.

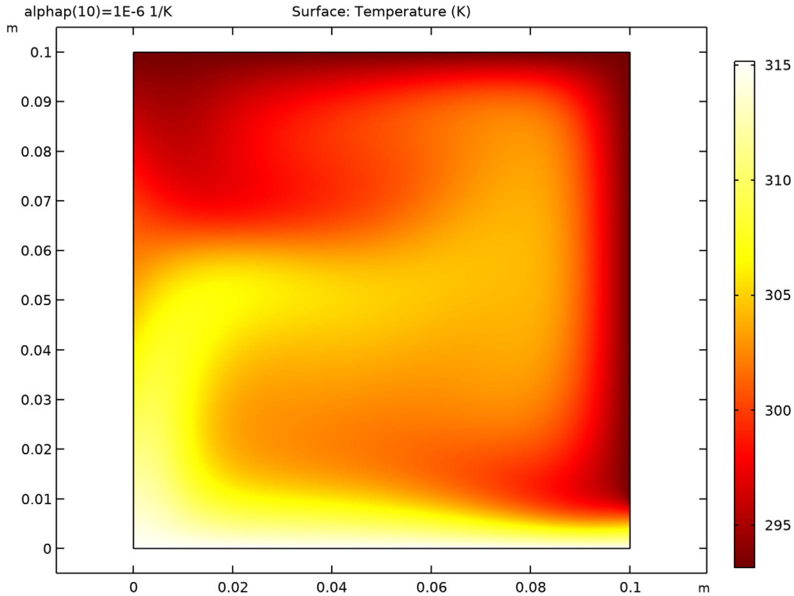

Figure 2: Temperature in a porous structure subjected to temperature gradients and subsequent free convection. The COMSOL Multiphysics simulation is in excellent agreement with published results from Ref. 1.

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

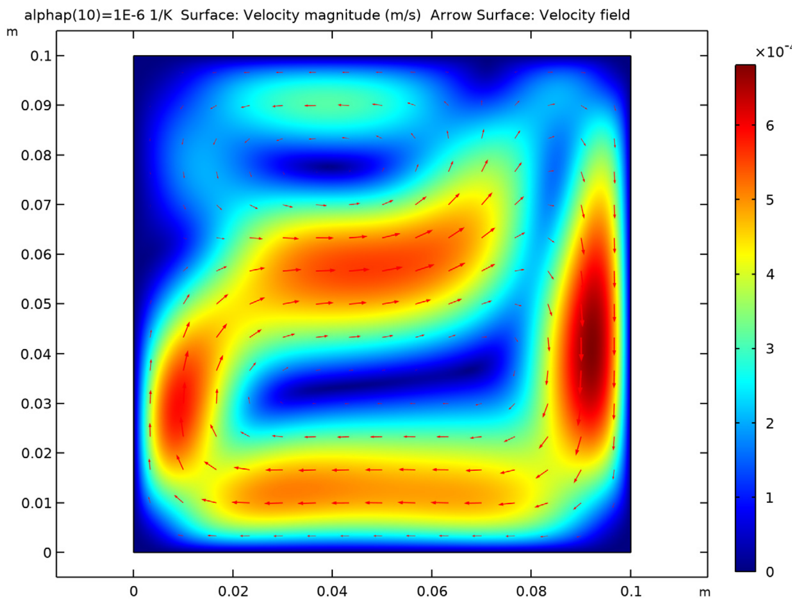

Figure 3: Velocity field for a Rayleigh number of Ra = 10^5.

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>Porous Media and Subsurface Flow>Brinkman Equations (br)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Porous Media (ht)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies>Stationary**.
8 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
rho	1000[kg/m^3]	1000 kg/m³	Fluid density
mu	0.001[Pa*s]	0.001 Pa·s	Fluid viscosity
alphap	1e-6[1/K]	1E-6 1/K	Fluid volumetric thermal expansion
kth	6[W/(m*K)]	6 W/(m·K)	Fluid thermal conductivity
gamma	1	1	Fluid ratio of specific heat
Cp	4200[J/(kg*K)]	4200 J/(kg·K)	Fluid heat capacity at constant pressure
epsilon	0.4	0.4	Porosity
kappa	1e-3[m^2]	0.001 m²	Permeability
p0	1[atm]	1.0133E5 Pa	Reference pressure
Tc	20[degC]	293.15 K	Reference temperature
Th	42[degC]	315.15 K	High temperature
L	0.1[m]	0.1 m	Length scale

Name	Expression	Value	Description
Pr	mu*Cp/kth	0.7	Prandtl number
Ra	Cp*rho^2*g_const*alphap*(Th-Tc)*L^3/(kth*mu)	1.5102E5	Rayleigh number

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 Click **Build Selected**.

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**.

BRINKMAN EQUATIONS (BR)

Fluid and Matrix Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Brinkman Equations (br)** click **Fluid and Matrix Properties 1**.
2 In the **Settings** window for **Fluid and Matrix Properties**, locate the **Fluid Properties** section.
3 From the ρ list, choose **User defined**. In the associated text field, type rho.
4 From the μ list, choose **User defined**. In the associated text field, type mu.
5 Locate the **Porous Matrix Properties** section. From the ε_p list, choose **User defined**. In the associated text field, type epsilon.
6 From the κ list, choose **User defined**. In the associated text field, type kappa.

Volume Force 1

1 In the **Physics** toolbar, click **Domains** and choose **Volume Force**.

Set up the Boussinesq buoyance term according to Equation 1.

2 In the **Settings** window for **Volume Force**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Volume Force** section. Specify the **F** vector as

0	x
rho*g_const*alphap*(T-Tc)	y

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**.

4 Click to collapse the **Discretization** section.

Fluid 1

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

2 In the **Settings** window for **Fluid**, locate the **Model Input** section.

3 From the p_A list, choose **User defined**. In the associated text field, type `p0`.

4 Locate the **Heat Convection** section. From the **u** list, choose **Velocity field (br)**.

5 Locate the **Heat Conduction, Fluid** section. From the k_f list, choose **User defined**. In the associated text field, type `kth`.

6 Locate the **Thermodynamics, Fluid** section. From the ρ_f list, choose **User defined**. In the associated text field, type `rho`.

7 From the $C_{p,\mathrm{f}}$ list, choose **User defined**. In the associated text field, type `Cp`.

8 From the γ list, choose **User defined**. In the associated text field, type `gamma`.

Porous Matrix 1

1 In the **Model Builder** window, click **Porous Matrix 1**.

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

3 From the ε_p list, choose **User defined**. In the associated text field, type `epsilon`.

4 Locate the **Heat Conduction, Porous Matrix** section. From the k_b list, choose **User defined**. Locate the **Thermodynamics, Porous Matrix** section. From the ρ_b list, choose **User defined**. From the $C_{p,b}$ list, choose **User defined**.

Initial Values 1

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

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 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
alphap (Fluid volumetric thermal expansion)	`0 1e-12 1e-11 1e-10 1e-9 1e-8 1e-7 6e-7 8e-7 1e-6`	`1/K`

6 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).

Isothermal Contours (ht)

The fourth default plot group shows the temperature contours.