Magnetic Drug Targeting in Cancer Therapy

Introduction

Current research on methods to target chemotherapy drugs in the human body includes the investigation of bio-compatible magnetic nanocarrier systems. For example, magnetic liquids such as ferrofluids can play an important role as drug carriers in the human body (Ref. 1). As such, they can be used for drug targeting in modern locoregional cancer treatment. A remaining challenge for this medical application is the choice of clinical setting. Important parameters are optimal adjustment of the external magnetic field and the choice of ferrofluid properties.

Avoiding damage to healthy human cells from chemotherapy drugs imposes an upper limit in the treatment dose. This limit impedes the chances of successful treatment of the tumor cells. One objective of modern cancer research is therefore to concentrate chemotherapy drugs locally on tumor tissue and to weaken the global exposure to the organism.

This model of the ferrohydrodynamics of blood demonstrates a simple setup for investigating an external magnetic field and its interaction with blood flow containing a magnetic carrier substance. The model treats the liquid as a continuum, which is a good first step. You can extend this model by particle tracing, making it a multiscale model. The equations and theory are based on Maxwell's equations and the Navier-Stokes equations. You first solve Maxwell's equations in the full modeling domain formed by permanent-magnet, blood-vessel, tissue, and air domains. A magnetic volume force then couples the resulting magnetic field to a fluid-flow problem in the blood-vessel domain described by the Navier-Stokes equations.

Model Definition[1]

The model geometry represents a blood vessel, a permanent magnet, surrounding tissue, and air in 2D. Blood feeds into the vessel from the left in Figure 1. The velocity and pressure fields are calculated in the blood stream. COMSOL Multiphysics computes the magnetic field (magnetic vector potential) generated by the permanent

1. This model was provided by Dr. Daniel J. Strauss, The Institute for New Materials, Inc., www.inm-gmbh.de.

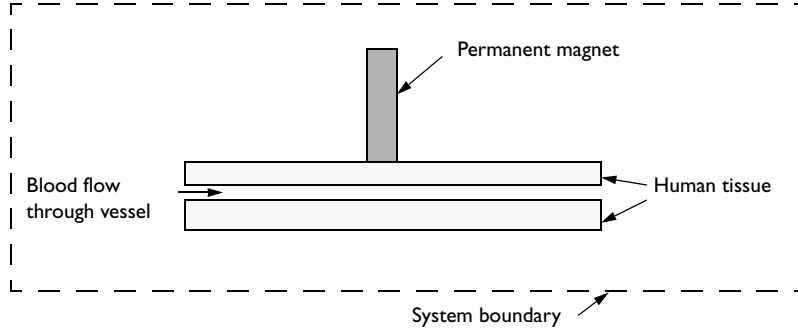magnet. This magnetic field generates a magnetic volume force that affects the flow field in the blood vessel.

Figure 1: Geometric representation of the model.

MAGNETOSTATIC EQUATIONS

Because the magnetic part of this problem is static, Maxwell-Ampere's law for the magnetic field $\mathbf{H}$ (A/m) and the current density $\mathbf{J}$ (A/m^2) applies:

$$\nabla \times \mathbf{H} = \mathbf{J} \tag{1}$$

Furthermore, Gauss' law for the magnetic flux density $\mathbf{B}$ (Vs/m^2) states that

$$\nabla \cdot \mathbf{B} = 0 \ . \tag{2}$$

The constitutive equations describing the relation between $\mathbf{B}$ and $\mathbf{H}$ in the different parts of the modeling domain read:

$$\mathbf{B} = \begin{cases} \mu_0 \mu_{r, \text{mag}} \mathbf{H} + \mathbf{B}_{\text{rem}} & \text{permanent magnet} \\ \mu_0 (\mathbf{H} + \mathbf{M}_{\text{ff}}(\mathbf{H})) & \text{blood stream} \\ \mu_0 \mathbf{H} & \text{tissue and air} \end{cases} \tag{3}$$

Here μ_0 is the magnetic permeability of vacuum (Vs/(A·m)); μ_r is the relative magnetic permeability of the permanent magnet (dimensionless); $\mathbf{B}_{\text{rem}}$ is the remanent magnetic flux (A/m); and $\mathbf{M}_{\text{ff}}$ is the magnetization vector in the blood stream (A/m), which is a function of the magnetic field, $\mathbf{H}$.

Defining a magnetic vector potential $\mathbf{A}$ such that

$$\mathbf{B} = \nabla \times \mathbf{A}, \qquad \nabla \cdot \mathbf{A} = 0 \ , \tag{4}$$

you finally, by substitution in Equation 1 through Equation 3, arrive at the following vector equation to solve:

$$\nabla \times \left(\frac{1}{\mu} \nabla \times \mathbf{A} - \mathbf{M} \right) = \mathbf{J}$$

Simplifying to a 2D problem with no perpendicular currents, this equation reduces to

$$\nabla \times \left(\frac{1}{\mu_0} \nabla \times \mathbf{A} - \mathbf{M} \right) = \mathbf{0} \ . \tag{5}$$

Note that this equation assumes that the magnetic vector potential has a nonzero component only perpendicularly to the plane, $\mathbf{A} = (0, 0, A_z)$.

An arc tangent expression with two material parameters α (A/m) and β (m/A) characterizes the induced magnetization $\mathbf{M}_{ff}(x, y) = (M_{ffx}, M_{ffy})$ of a ferrofluid (Ref. 2):

$$M_x = \alpha \operatorname{atan} \left(\frac{\beta}{\mu_0} \frac{\partial A_z}{\partial y} \right)$$

$$M_y = \alpha \operatorname{atan} \left(\frac{\beta}{\mu_0} \frac{\partial A_z}{\partial x} \right)$$

For the magnetic fields of interest, it is possible to linearize these expressions to obtain

$$M_x = \frac{\chi}{\mu_0} \frac{\partial A_z}{\partial y}$$

$$M_y = -\frac{\chi}{\mu_0} \frac{\partial A_z}{\partial x} \tag{6}$$

where $\chi = \alpha \beta$ is the magnetic susceptibility.

Boundary Conditions

Along a system boundary reasonably far away from the magnet (see Figure 1) you can apply a magnetic insulation boundary condition, $A_z = 0$.

FLUID FLOW EQUATIONS

The Navier-Stokes equations describe the time-dependent mass and momentum balances for an incompressible flow:

$$\rho\frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot \eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) + \rho \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p = \mathbf{F} \tag{7}$$

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

where η denotes the dynamic viscosity (kg/(m·s)), $\mathbf{u}$ the velocity (m/s), ρ the fluid density (kg/m^3), p the pressure (N/m^2), and $\mathbf{F}$ a volume force (N/m^3).

With the assumption that the magnetic nanoparticles in the fluid do not interact, the magnetic force $\mathbf{F} = (F_x, F_y)$ on the ferrofluid for relatively weak fields is given by $\mathbf{F} = |\mathbf{M}|\nabla|\mathbf{H}|$. Using Equation 3, Equation 4, and Equation 6 then leads to the expressions

$$F_x = \frac{\chi}{\mu_0 \mu_r^2}\left(\frac{\partial A_z}{\partial x}\frac{\partial^2 A_z}{\partial x^2} + \frac{\partial A_z}{\partial y}\frac{\partial^2 A_z}{\partial x \partial y}\right)$$

$$F_y = \frac{\chi}{\mu_0 \mu_r^2}\left(\frac{\partial A_z}{\partial x}\frac{\partial^2 A_z}{\partial x \partial y} + \frac{\partial A_z}{\partial y}\frac{\partial^2 A_z}{\partial y^2}\right)$$

To get the final expression for the volume force in the blood stream, multiply these expressions by the ferrofluid mass fraction, k_{ff}.

The non-Newtonian aspect of blood flow is significant for low velocities (and if this is taken together with the fact that there are times in a cardiac cycle where the flow rate is near zero), an appropriate model for viscosity changes must be taken into consideration. The use of the Generalized Power Law model (Ref. 4) approximates the wall shear stress better than the Newtonian model for low inlet velocities and in regions of low shear, closely agrees with the Carreau model for low to mid-range shear and is effectively Newtonian at mid-range to high shear. This model defines the dynamic viscosity as

$$\mu = 0.1\lambda|\dot{\gamma}|^{n-1} \tag{8}$$

with

$$\lambda(\dot{\gamma}) = \mu_\infty + \Delta\mu \exp\left[-\left(1 + \frac{|\dot{\gamma}|}{a}\right)\exp\left(-\frac{b}{|\dot{\gamma}|}\right)\right]$$

$$n(\dot{\gamma}) = n_\infty + \Delta n \exp\left[-\left(1 + \frac{|\dot{\gamma}|}{c}\right)\exp\left(-\frac{d}{|\dot{\gamma}|}\right)\right]$$

where $\dot{\gamma}$ is the shear rate.

Boundary Conditions

On the vessel walls, apply no-slip conditions, $u = v = 0$. At the outlet, you can set an outlet pressure condition, $p = 0$. At the inlet boundary, specify a parabolic flow profile on the normal inflow velocity according to $4U_m s(1-s)$, where s is a boundary segment length parameter that goes from 0 to 1 along the inlet boundary segment and U_m is the maximal flow velocity. To emulate the heart beat, the inflow velocity follows a sinusoidal expression in time:

$$U_0 = 2U_m s(1-s)(\sin(\omega t) + \sqrt{\sin(\omega t)^2})$$

Selecting the angular velocity ω to be 2π rad/s gives a heart beat rate of 60 beats per minute. Figure 2 displays the resulting expression (normalized to unity).
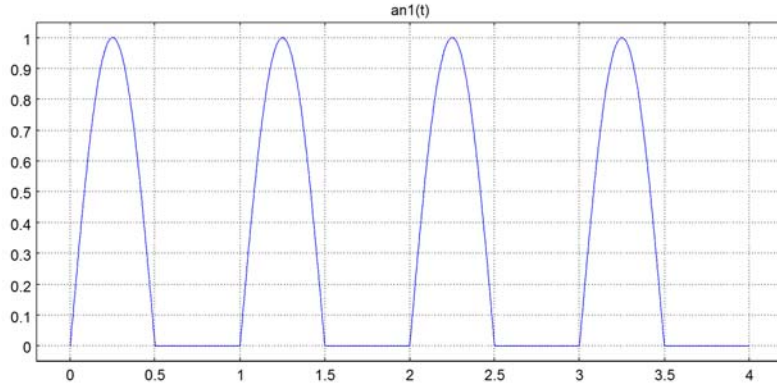

Figure 2: Simulated heart beat.

Model Data

Table 8-1 lists the relevant material properties for the model.

TABLE 8-1: MODEL DATA

QUANTITY	DESCRIPTION	VALUE
$\mu_{r,mag}$	Relative permeability, magnet	$5 \cdot 10^3$
B_{rem}	Remanent flux density, magnet	0.5 T
α	Ferrofluid magnetization-curve parameter	10^{-4} A/m
β	Ferrofluid magnetization-curve parameter	$3 \cdot 10^{-5}$ (A/m)$^{-1}$
ρ	Density, blood	1060 kg/m^3
μ	Dynamic viscosity, blood	$3.5 \cdot 10^{-3}$ kg/(m·s)

Results and Discussion

Figure 3 shows a detail from the plot of the magnetic field strength. The highest B-field strength clearly occurs inside the magnet. To see the low-level variations in the surrounding tissue and vessels, the plot does not show magnetic flux densities above 0.17 T. The geometric form of the magnet generates strong fields just outside of the rounded corners. Sharper corners generate even stronger local fields.
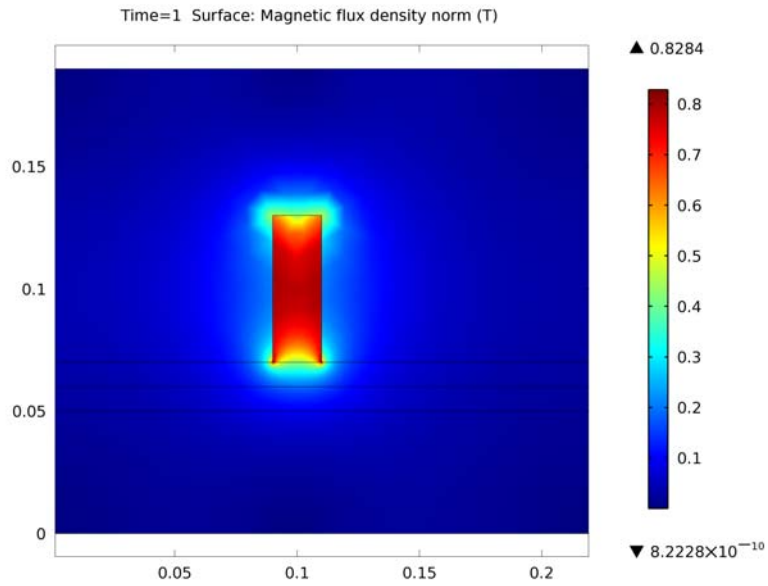
Figure 3: Magnetic vector potential and magnetic flux density, B field

Figure 4 reveals the velocity field between two heart beats, where the net throughput is zero.

Time=1 Surface: Velocity magnitude (m/s)
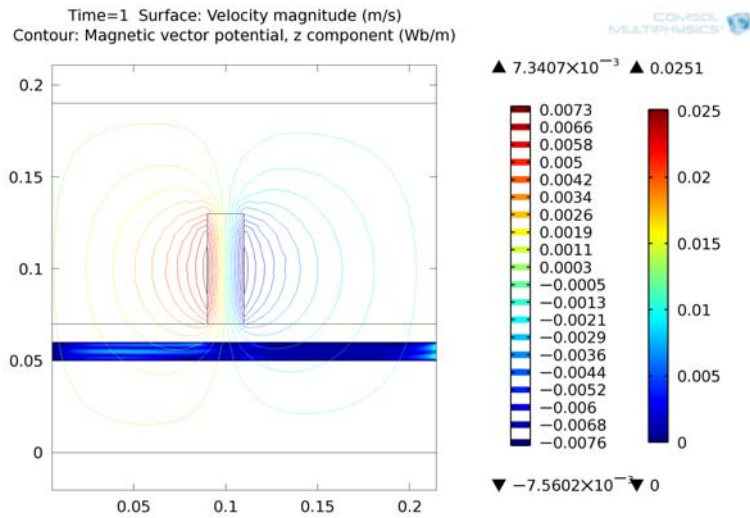Contour: Magnetic vector potential, z component (Wb/m)

Figure 4: Velocity field at zero blood throughput (t = 1).

Figure 5 shows the induced fluid rotation due to the magnetic field between two heart beats.
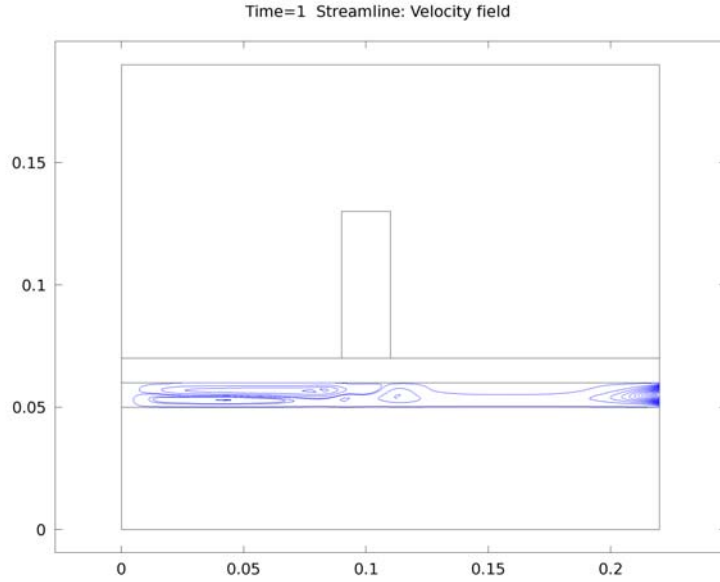
Time=1 Streamline: Velocity field

Figure 5: Streamline at zero blood throughput (t = 1).

The viscosity determined using the non-Newtonian fluid model can be compared to the Newtonian viscosity using the non-Newtonian importance factor (NNIF) shown in equation 9 and plotted on Figure 6 (defined by Johnston et. al in Ref. 4). Here, μ_{NN} is the non-Newtonian viscosity and is μ_N the Newtonian viscosity.

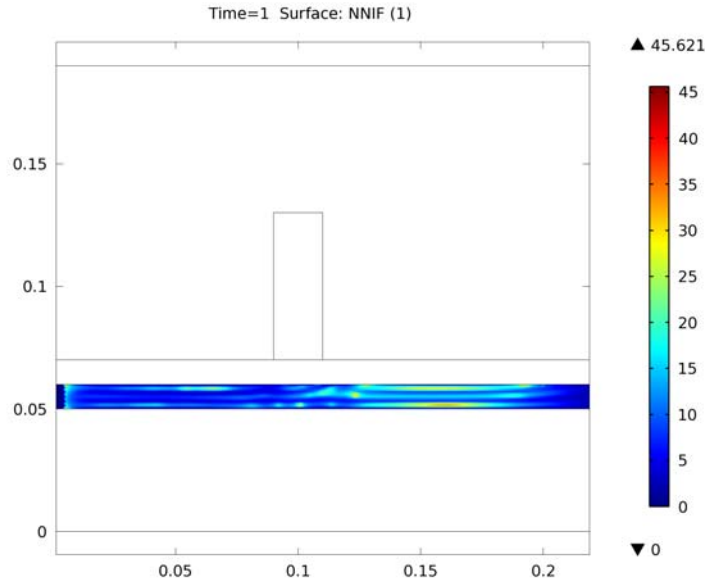$$NNIF = \frac{\mu_{NN}}{\mu_N} \tag{9}$$

Figure 6: Non Newtonian importance factor at zero blood throughput (t =1).

Notes About the COMSOL Implementation

EQUATIONS

To model Equation 5, use the Magnetic fields (mf) application mode in 2D.

STAGED SOLUTION

Because the magnetostatic problem is a stationary nonlinear problem that is independent of the fluid-flow problem, you need to solve that only once. You can therefore start by solving only Equation 5 with the stationary solver. Then proceed with solving only the fluid-flow problem, Equation 7, with the static magnetic potential as input. Solve the fluid-flow problem using the time-dependent solver.

References

1. P.A. Voltairas, D.I. Fotiadis, and L.K. Michalis, "Hydrodynamics of Magnetic Drug Targeting," *J. Biomech.*, vol. 35, pp. 813–821, 2002.

2. C.M. Oldenburg, S.E. Borglin, and G.J. Moridis, "Numerical Simulation of Ferrofluid Flow for Subsurface Environmental Engineering Applications," *Transport in Porous Media*, vol. 38, pp. 319–344, 2000.

3. R.E. Rosensweig, *Ferrohydrodynamics*, Dover Publications, New York, 1997.

4. B.M. Johnston, P. R. Johnston, S. Corney, and D. Kilpatrick, "Non-Newtonian Blood Flow in Human Right Coronary Arteries: Steady State Simulations," *Journal of Biomechanics*, vol. 37, pp. 709-720, 2004

Model Library path: COMSOL_Multiphysics/Multiphysics/ magnetic_drug_targeting

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D** button.

3 Click **Next**.

4 In the **Add Physics** tree, select **AC/DC>Magnetic Fields (mf)**.

5 Click **Add Selected**.

6 In the **Add Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

7 Click **Add Selected**.

8 Clear the selection in the **Selected physics** tree.

9 Click **Next**.

10 In the **Studies** tree, select **Preset Studies for Selected Physics>Stationary**.

11 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 Go to the **Settings** window for Parameters.

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

NAME	EXPRESSION	DESCRIPTION
mur_mag	1	Relative permeability, magnet
B_rem	1[T]	Remanent flux density, magnet
chi_ff	0.3	Magnetic susceptibility, ferrofluid
mur_ff	1+chi_ff	Relative permeability, ferrofluid
k_ff	0.1	Ferrofluid mass fraction in blood stream
rho	1060[kg/m^3]	Density of blood
U_m	50[cm/s]	Maximum flow velocity
f	60[1/min]	Heart-beat rate
omega	2*pi[rad]*f	Pulse angular velocity
mufluid	0.0035[Pa*s]	Newtonian viscosity of blood

Analytic 1 (an1)

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Analytic**.

2 Go to the **Settings** window for Analytic.

3 Locate the **Parameters** section. In the **Expression** edit field, type
   U_m*(sin(omega*t)+sqrt(sin(omega*t)^2)).

4 In the **Arguments** edit field, type t.

5 Click to expand the **Plot Parameters** section.

6 In the **[[plotargs]]** table, enter the following settings:

LOWER LIMIT	UPPER LIMIT
0	4

7 Click the **Plot** button.

Step 1 (step1)

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Step**.

2 Go to the **Settings** window for Step.

3 Locate the **Parameters** section. In the **Location** edit field, type 0.01.

4 Click to expand the **Smoothing** section.

5 In the **Size of transition zone** edit field, type 0.02.

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Model Builder** window, right-click **Model 1 (mod1)>Geometry 1** and choose **Rectangle**.

2 Go to the **Settings** window for Rectangle.

3 Locate the **Size** section. In the **Width** edit field, type 0.02.

4 In the **Height** edit field, type 0.06.

5 Locate the **Position** section. In the **x** edit field, type 0.09.

**6** In the **y** edit field, type 0.07.

Rectangle 2 (r2)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.

2 Go to the **Settings** window for Rectangle.

3 Locate the **Size** section. In the **Width** edit field, type 0.22.

4 In the **Height** edit field, type 0.05.

Rectangle 3 (r3)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.

2 Go to the **Settings** window for Rectangle.

3 Locate the **Size** section. In the **Width** edit field, type 0.22.

4 In the **Height** edit field, type 0.02.

5 Locate the **Position** section. In the **y** edit field, type 0.05.

Rectangle 4 (r4)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.

2 Go to the **Settings** window for Rectangle.

3 Locate the **Size** section. In the **Width** edit field, type 0.22.

4 In the **Height** edit field, type 0.13.

5 Locate the **Position** section. In the **y** edit field, type 0.06.

6 Click the **Build All** button.

7 Click the **Zoom Extents** button on the Graphics toolbar.
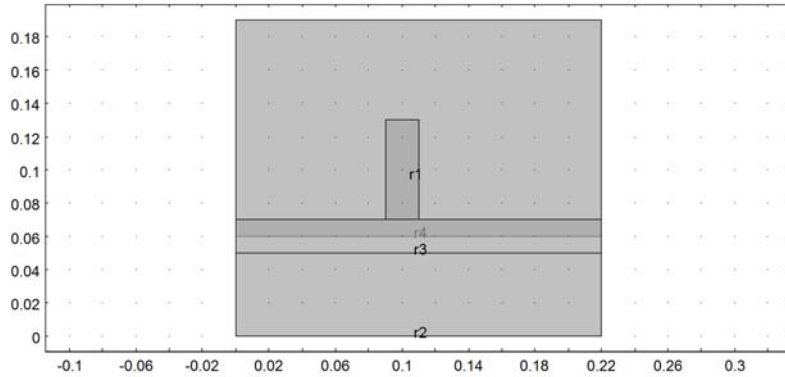
DEFINITIONS

Variables 1

1 In the **Model Builder** window, right-click **Model 1 (mod1)>Definitions** and choose **Variables**.

2 Go to the **Settings** window for Variables.

3 Locate the **Geometric Scope** section. From the **Geometric entity level** list, select **Domain**.

4 Select Domain 2 only.

5 Go to the **Settings** window for Variables.

6 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
mu0	mu0_const	Permeability of vacuum
M_ffx	k_ff*(chi_ff/mu0)*Azy	Induced ferrofluid magnetization, x-component
M_ffy	-k_ff*(chi_ff/ mu0)*Azx	Induced ferrofluid magnetization, y-component

NAME	EXPRESSION	DESCRIPTION
F_ffx	k_ff*(Azx*Azxx+Azy*Az xy)*chi_ff/ (muO*mur_ff^2)	Ferrofluid volume force, x-component
F_ffy	k_ff*(Azx*Azxy+Azy*Az yy)*chi_ff/ (muO*mur_ff^2)	Ferrofluid volume force, y-component

Variables 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 Go to the **Settings** window for Variables.

3 Locate the **Geometric Scope** section. From the **Geometric entity level** list, select **Domain**.

4 Select Domain 2 only.

5 Go to the **Settings** window for Variables.

6 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
muinf	0.035	
ninf	1.0	
dmu	0.25	
dn	0.45	
a	50	
b	3	
c	50	
d	4	
lambda	muinf+dmu*exp(-(1+abs(spf.sr)/a)*exp(-b/abs(spf.sr)))	
npow	ninf-dn*exp(-(1+abs(spf.sr)/ c)*exp(-d/abs(spf.sr)))	
mufluidN	1e-1*(lambda*abs(spf.sr)^(np ow-1))	Non-Newtonian viscosity of blood in SI units

MAGNETIC FIELDS (MF)

In the **Model Builder** window, expand the **Model 1 (mod1)>Magnetic Fields (mf)** node.

Ampère's Law 2

1 Right-click **Magnetic Fields (mf)** and choose **Ampère's Law**.

2 Select Domains 1, 3, and 4 only.

3 Go to the **Settings** window for Ampère's Law.

4 Locate the **Magnetic Field** section. From the μ_r list, select **User defined**. Locate the **Conduction Current** section. From the σ list, select **User defined**. Locate the **Electric Field** section. From the ϵ_r list, select **User defined**.

Ampère's Law 3

1 In the **Model Builder** window, right-click **Magnetic Fields (mf)** and choose **Ampère's Law**.

2 Select Domain 2 only.

3 Go to the **Settings** window for Ampère's Law.

4 Locate the **Magnetic Field** section. From the **Constitutive relation** list, select **Magnetization**.

5 Specify the **M** vector as

M_ffx	x
M_ffy	y
0	z

6 Locate the **Conduction Current** section. From the σ list, select **User defined**. Locate the **Electric Field** section. From the ϵ_r list, select **User defined**.

Ampère's Law 4

1 In the **Model Builder** window, right-click **Magnetic Fields (mf)** and choose **Ampère's Law**.

2 Select Domain 5 only.

3 Go to the **Settings** window for Ampère's Law.

4 Locate the **Magnetic Field** section. From the **Constitutive relation** list, select **Remanent flux density**.

5 From the μ_r list, select **User defined**. From the **[[mur_matrixbox-1]]** list, select **Diagonal**.

6 In the μ_r table, enter the following settings:

mur_mag	0	0
0	mur_mag	0
0	0	1

7 Specify the $\mathbf{B_r}$ vector as

0	x
B_rem	y
0	z

8 Locate the **Conduction Current** section. From the σ list, select **User defined**. Locate the **Electric Field** section. From the ε_r list, select **User defined**.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, click **Model 1 (mod1)>Laminar Flow (spf)**.

2 Go to the **Settings** window for Laminar Flow.

3 Locate the **Domains** section. Click **Clear Selection**.

4 Select Domain 2 only.

5 Go to the **Settings** window for Laminar Flow.

6 Locate the **Physical Model** section. From the **Compressibility** list, select **Incompressible flow**.

Fluid Properties 1

1 In the **Model Builder** window, expand the **Laminar Flow (spf)** node, then click **Fluid Properties 1**.

2 Go to the **Settings** window for Fluid Properties.

3 Locate the **Fluid Properties** section. From the ρ list, select **User defined**. In the associated edit field, type rho.

The use of a step function will allow to attain a good initial guess for the shear strain by momentarily solving a Newtonian flow. After 0.02s the flow becomes completely Non-Newtonian

4 From the μ list, select **User defined**. In the associated edit field, type
`mufluid+(mufluidN-mufluid)*step1(t)`.

Volume Force 1

1 In the **Model Builder** window, right-click **Laminar Flow (spf)** and choose **Volume Force**.

2 Select Domain 2 only.

3 Go to the **Settings** window for Volume Force.

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

F_ffx	x
F_ffy	y

Inlet 1

1 In the **Model Builder** window, right-click **Laminar Flow (spf)** and choose **Inlet**.

2 Select Boundary 3 only.

3 Go to the **Settings** window for Inlet.

4 Locate the **Boundary Condition** section. In the U_0 edit field, type
`2*s*(1-s)*an1(t[1/s])`.

Outlet 1

1 In the **Model Builder** window, right-click **Laminar Flow (spf)** and choose **Outlet**.

2 Select Boundary 16 only.

MESH 1

Size

1 In the **Model Builder** window, right-click **Model 1 (mod1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

2 In the **Model Builder** window, click **Size**.

3 Go to the **Settings** window for Size.

4 Locate the **Element Size** section. From the **Predefined** list, select **Extremely fine**.

Two solutions steps are taken in this model. Use the stationary solver to solve for magnetic field physics. Next, use of a transient solver for the laminar flow physics. COMSOL automatically stores the previous solution for later use. Finally make use of a tighter time stepping tolerance

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

2 Go to the **Settings** window for Stationary.

3 Locate the **Physics Selection** section. In the **Physics interfaces** list, select **Laminar flow (spf)**.

4 Clear the **Use in this study** check box.

Step 2: Time Dependent

1 In the **Model Builder** window, right-click **Study 1** and choose **Time Dependent**.

2 Go to the **Settings** window for Time Dependent.

3 Locate the **Study Settings** section. Click the **Range** button.

4 Go to the **Range** dialog box.

5 In the **Stop** edit field, type 2.

6 In the **Step** edit field, type 0.02.

7 Click the **Replace** button.

8 Go to the **Settings** window for Time Dependent.

9 Locate the **Study Settings** section. Select the **Relative tolerance** check box.

10 In the associated edit field, type 0.001.

11 Locate the **Physics Selection** section. Clear the **Use in this study** check box.

12 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

2D Plot Group 1

1 Go to the **Settings** window for 2D Plot Group.

2 Locate the **Data** section. From the **Time** list, select **1**.

3 Click the **Plot** button.

2D Plot Group 2

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

2 Go to the **Settings** window for 2D Plot Group.

3 Locate the **Data** section. From the **Time** list, select **1**.

4 Right-click **Results>2D Plot Group 2** and choose **Surface**.

5 Go to the **Settings** window for Surface.

6 In the upper-right corner of the **Expression** section, click **Replace Expression**.

7 From the menu, choose **Laminar Flow>Velocity magnitude (spf.U)**.

8 In the **Model Builder** window, right-click **2D Plot Group 2** and choose **Contour**.

9 Go to the **Settings** window for Contour.

10 In the upper-right corner of the **Expression** section, click **Replace Expression**.

11 From the menu, choose **Magnetic Fields>Magnetic vector potential>Magnetic vector potential, z component (Az)**.

12 Click the **Plot** button.

13 Click the **Zoom Extents** button on the Graphics toolbar.

2D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

2 Go to the **Settings** window for 2D Plot Group.

3 Locate the **Data** section. From the **Time** list, select **1**.

4 Right-click **Results>2D Plot Group 3** and choose **Streamline**.

5 Go to the **Settings** window for Streamline.

6 In the upper-right corner of the **Expression** section, click **Replace Expression**.

7 From the menu, choose **Laminar Flow>Velocity field (u, v)**.

8 Locate the **Streamline Positioning** section. From the **Positioning** list, select **Magnitude controlled**.

9 Locate the **Coloring and Style** section. From the **Color** list, select **Blue**.

10 Click the **Plot** button.

2D Plot Group 4

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

2 Go to the **Settings** window for 2D Plot Group.

3 Locate the **Data** section. From the **Time** list, select **1**.

4 Right-click **Results>2D Plot Group 4** and choose **Surface**.

5 Go to the **Settings** window for Surface.

6 Locate the **Expression** section. In the **Expression** edit field, type `(spf.mu/mufluid)*(x>0.005)`.

7 Select the **Description** check box.

8 In the associated edit field, type `NNIF`.

9 Click the **Plot** button.