COMSOL MULTIPHYSICS®

Electromagnetic Forces on Parallel Current-Carrying Wires

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

© COPYRIGHT 2008. All right reserved. No part of this documentation may be photocopied or reproduced in any form without prior written consent from COMSOL AB. COMSOL, COMSOL Multiphysics, COMSOL Reaction Engineering Lab, and FEMLAB are registered trademarks of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

COMSOL

Electromagnetic Forces on Parallel Current-Carrying Wires

Introduction

One ampere is defined as the constant current in two straight parallel conductors of infinite length and negligible circular cross section, placed one meter apart in vacuum, that produces a force of $2 \cdot 10^{-7}$ newton per meter of length (N/m). This model shows a setup of two parallel wires in the spirit of this definition, but with the difference that the wires have finite cross sections.

For wires with circular cross section carrying a uniform current density as in this example, the mutual magnetic force will be the same as for line currents. This can be understood by the following arguments: Start from a situation where both wires are line currents ($\mathbf{I}$). Each line current will be subject to a Lorentz force ($\mathbf{I} \times \mathbf{B}$) where the magnetic flux density ($\mathbf{B}$) is the one produced by the other wire. Now, give one wire a finite radius. It follows directly from circular symmetry and Maxwell-Ampère's law that, outside this wire, the produced flux density is exactly the same as before so the force on the remaining line current is unaltered. Further, the net force on the wire with the distributed current density must be of exactly the same magnitude (but with opposite direction) as the force on the line current so that force did not change either. If the two wires exchange places, the forces must still be the same and it follows from symmetry that the force is independent of wire radius as long as the wire cross sections do not intersect. The wires can even be cylindrical shells or any other shape with circular symmetry of the cross section that. For an experimental setup, negligible cross section is required as resistive voltage drop along the wires and Hall effect may cause electrostatic forces that increase with wire radius but such effects are not included in this model.

The force between the wires is computed using three different methods: by integrating the volume (Lorentz) force density over the wire cross section; by integrating the stress tensor on the boundary; and finally by using the principle of virtual work. The results converge to $2 \cdot 10^{-7}$ N/m for the 1 ampere definition, as expected.

Model Definition

The model is built using the 2D Perpendicular Induction Currents application mode. The modeling plane is a cross section of the two wires and the surrounding air.

DOMAIN EQUATIONS

The equation formulation in the 2D Perpendicular Induction Currents application mode assume that the only nonzero component of the magnetic vector potential is A_z. This corresponds to all currents being perpendicular to the modeling plane. The following equation is solved:

$$\nabla \times (\mu \nabla \times A_z) = J_z^e$$

where μ is the permeability of the medium and J_z^e is the externally applied current. J_z^e is set so that the applied current in the wires equals 1 A, but with different signs.

BOUNDARY CONDITIONS

At the exterior boundary of the air domain, A_z, is fixed to zero. The interior boundaries between the wires and the air only assume continuity, corresponding to a homogeneous Neumann condition.

Results and Discussion

The expression for the surface stress reads

$$\mathbf{n}_1 T_2 = -\frac{1}{2}(\mathbf{H} \cdot \mathbf{B})\mathbf{n}_1 + (\mathbf{n}_1 \cdot \mathbf{H})\mathbf{B}^T$$

where $\mathbf{n}_1$ is the boundary normal pointing out from the conductor wire and T_2 the stress tensor of air. The closed line integral of this expression around the circumference of either wire evaluates to $-1.99 \cdot 10^{-7}$ N/m. The minus sign indicates that the force between the wires is repulsive.

The volume force density is given by

$$\mathbf{F} = \mathbf{J} \times \mathbf{B} = \left[-J_z^e \cdot B_y, J_z^e \cdot B_x, 0\right]$$

The surface integral of the x component of the volume force on the cross section of a wire gives the result $-1.99 \cdot 10^{-7}$ N/m.

The model also computes the electromagnetic force by using the principle of virtual work. It adds the application modes for parameterized geometry and sensitivity to introduce the displacement and differentiate the energy with respect to the displacement according to

$$\mathbf{F} = \frac{\delta W}{\delta \mathbf{x}} \tag{1}$$

By refining the mesh and re-solving the problem, you can verify that the solution indeed converges to $-2 \cdot 10^{-7}$ (N/m).

Model Library path: `ACDC_Module/Tutorial_Models/parallel_wires`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Start COMSOL Multiphysics.

2 In the **Model Navigator**, select **2D** from the **Space dimension** list.

3 From the **Application Modes** tree, select the node **AC/DC Module>Virtual Work>Virtual Work for Perpendicular Currents**.

4 Click **OK**.

OPTIONS AND SETTINGS

1 Choose **Options>Constants**.

2 In the **Constants** dialog box, enter the following names, expressions, and (optionally) descriptions:

NAME	EXPRESSION	DESCRIPTION
r	0.2[m]	Wire radius
A	pi*r^2	Wire area
I0	1[A]	Current through wire
J0	I0/A	Current density

The radius r of each wire is 0.2 m, and the cross-section area is $A = \pi r^2$. The total current is $I_0 = 1$ A and the current density is $J_0 = I_0/A$.

3 Click **OK**.

GEOMETRY MODELING

1 Press Shift, then click the **Ellipse/Circle (Centered)** button on the Draw toolbar.

2 In the **Circle** dialog box, specify a circle C1 with **Radius** `0.2` centered at (`-0.5`, `0`), then click **OK**.

3 In the same way, create circles C2, C3, and C4 according to the following table (for properties not mentioned, use the default settings):

SETTINGS	C2	C3	C4
Radius	0.2	1	1.5
X	0.5	0	0
Y	0	0	0

4 Click the **Zoom Extents** button on the Main toolbar.

This completes the geometry-modeling stage. The finished geometry on your screen should look like that in Figure 1.

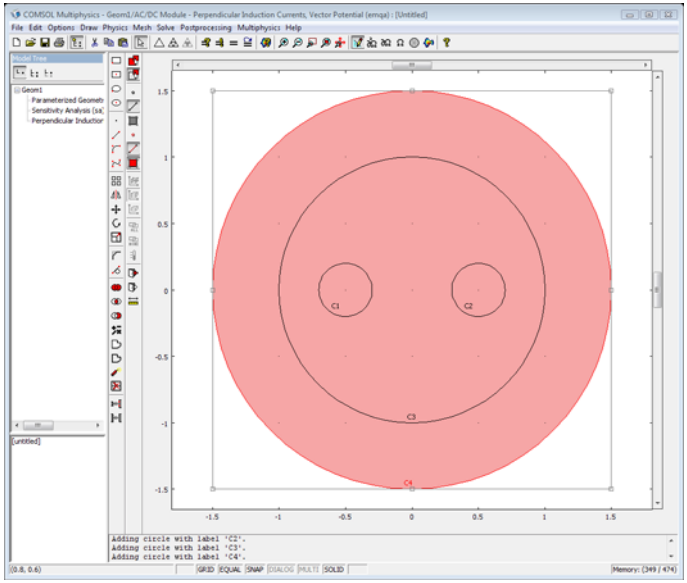


Figure 1: The model geometry.

PHYSICS SETTINGS

Subdomain Settings—Perpendicular Induction Currents

1 In the **Multiphysics** menu, make sure that **Perpendicular Induction Currents, Vector Potential (emqa)** is the active application mode.

2 From the **Physics** menu, choose **Subdomain Settings**.

3 On the **Physics** page, enter the following settings:

SETTINGS	SUBDOMAIN 3	SUBDOMAIN 4
J^e_z	J0	-J0

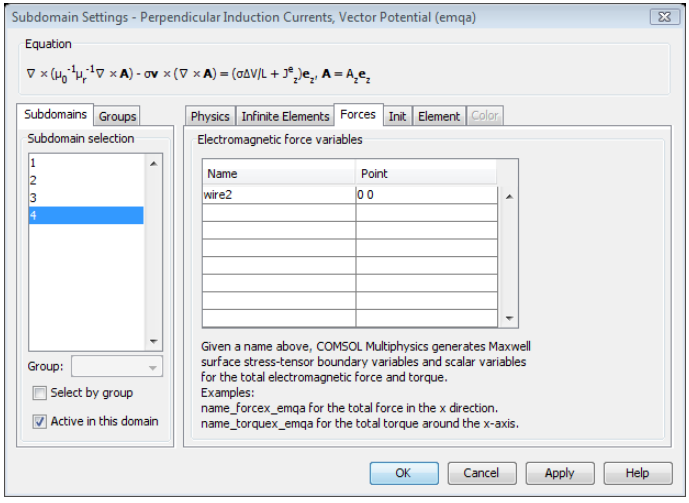
4 On the **Infinite Elements** page, enter the following settings:

SETTINGS	SUBDOMAIN 1
Type of Infinite Element	Cylindrical

5 On the **Forces** page, specify the following **Electromagnetic force variables**:

SETTINGS	SUBDOMAIN 3	SUBDOMAIN 4
Name	wire1	wire2
Point	0 0	0 0

Note that you do not need to enter the point coordinates, just press Enter after typing the name.



6 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions—Perpendicular Induction Currents

Use the default boundary conditions.

Subdomain Settings—Parameterized Geometry

1. From the **Multiphysics** menu, select **Parameterized Geometry (pg)**.
2. From the **Physics** menu, open the **Subdomain Settings** dialog box.
3. Select Subdomain 1 and click the **No displacement** option button.

   It is important that the inside of the infinite element domain is kept fixed. The formulation is invalid for a deformed circular domain.
4. Click **OK**.

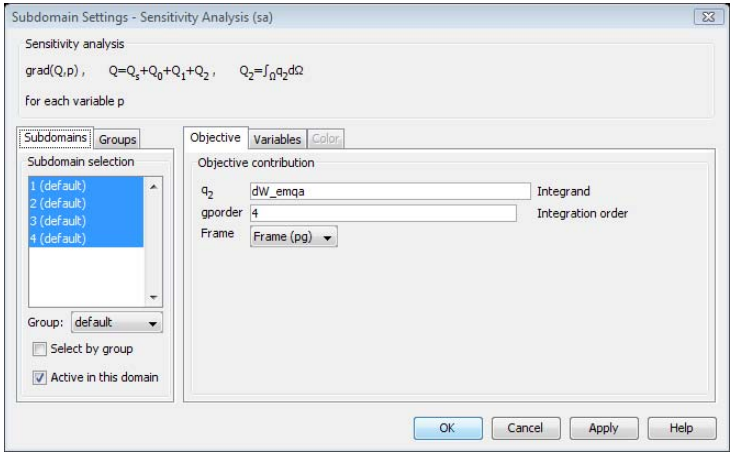
Boundary Conditions—Parameterized Geometry

1. From the **Physics** menu, open the **Boundary Settings** dialog box.
2. Select the **Interior boundaries** check box.
3. Select Boundaries 13–16 (the right wire), then select the group **fixed** from the **Group** list. Boundaries in this group remain fixed with no displacement.
4. Select Boundaries 5–8 (the left wire), then select **displacement** from the **Group** list. Boundaries in this group are displaced using the variables `deltaX` and `deltaY`, the dependent variables for the Sensitivity Analysis application mode.
5. Click **OK**.

Subdomain Settings—Sensitivity Analysis

When part of a Virtual Work predefined coupling, you do not need to specify any settings for the Sensitivity Analysis application mode. It can nevertheless be instructive to inspect the predefined coupling to the Perpendicular Induction Currents, Vector Potential application mode. However, if you want to, you can skip directly to the section "Mesh Generation" on page 8.

1. From the **Multiphysics** menu, select **Sensitivity Analysis (sa)**.
2. From the **Physics** menu, select **Subdomain Settings**.
3. Select all subdomains.
4. In the **Objective contribution** area on the **Objective** page, note that the $\mathbf{q_2}$ edit field contains the predefined entry `dW_emqa`, as the following figure also shows.



The quantity `dW_emqa` is the integrand for the total electromagnetic energy density. The objective function `Q_sa`—defined as the integral of all objective contributions at all domain levels—thus equals the total electromagnetic energy, W. Consequently, according to the principal of virtual work, the sensitivities `deltaXt` and `deltaYt` encode the force components F_x and F_y; see Equation 1 on page 3.

MESH GENERATION

Click the **Initialize Mesh** button on the Main toolbar to initialize the mesh.

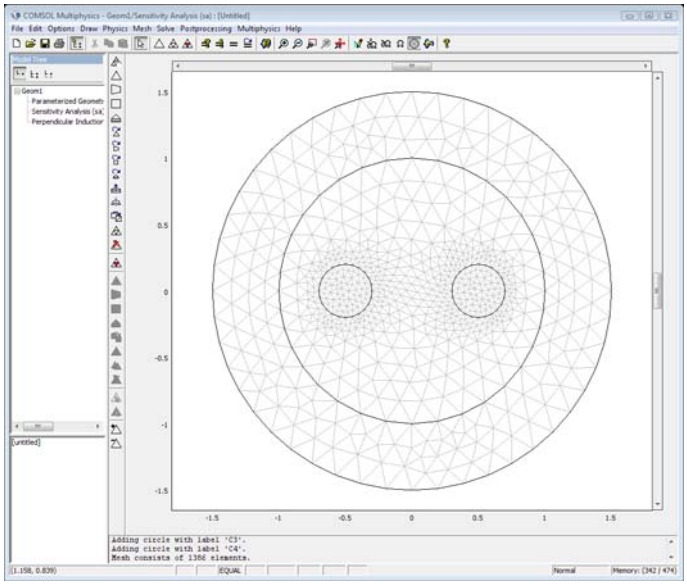


Figure 2: The mesh.

COMPUTING THE SOLUTION

1. From the **Solve** menu, open the **Solver Parameters** dialog box.
2. From the **Linear system solver** list on the **General** page, select **Direct (PARDISO)**.

   Note that the **Optimization/Sensitivity** check box is selected. On the **Optimization/Sensitivity** page you should also be able to verify that the **Design variables** edit field contains the entry `deltaX deltaY`.
3. Click **OK**.
4. Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows the x-displacement from the parameterized geometry, which is almost zero everywhere. Follow these steps to visualize the magnitude of the volume force density:

1. Click the **Plot Parameters** button on the Main toolbar.

2 Click the **Surface** tab. In the **Expression** edit field, type `sqrt(FLtzx_emqa^2+FLtzy_emqa^2)` in order to plot the magnitude of the Lorentz force distribution.

3 On the **Contour** page, select the **Contour plot** check box.

4 From the **Predefined quantities** list, select **Perpendicular Induction Currents, Vector Potential (emqa)>Magnetic field, norm**.

5 Click **Apply** to generate the plot.

6 To zoom in on the wires, choose **Axes/Grid Settings** from the **Options** menu. On the **Axis** page, type `-1` in the **x min** and **y min** edit fields, and type `1` in the **x max** and **y max** edit fields. Make sure that the **Axis equal** check box is selected, then click **Apply**.

Figure 3 shows the result.

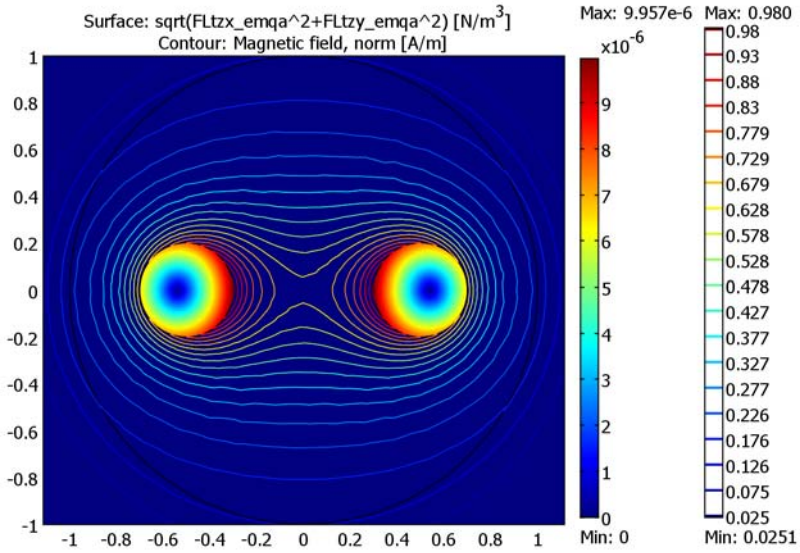


Figure 3: Force density and magnetic field.

To visualize the air stress tensor on the boundary, proceed as follows:

1 Return to the **Contour** page of the **Plot Parameters** dialog box and clear the **Contour plot** check box.

2 On the **Arrow** page, select the **Arrow plot** check box and select **Boundaries** from the **Plot arrows on** list.

3 On the **Boundary Data** page, select **Perpendicular Induction Currents, Vector Potential (emqa)>Maxwell surface stress tensor (wire1)** from the **Predefined quantities** list.

4 In the **Arrow parameters** area, clear the **Auto** check box for **Scale factor**, then type `0.1` in the associated edit field.

5 Click **OK** to close the dialog box and generate the plot shown in Figure 4.

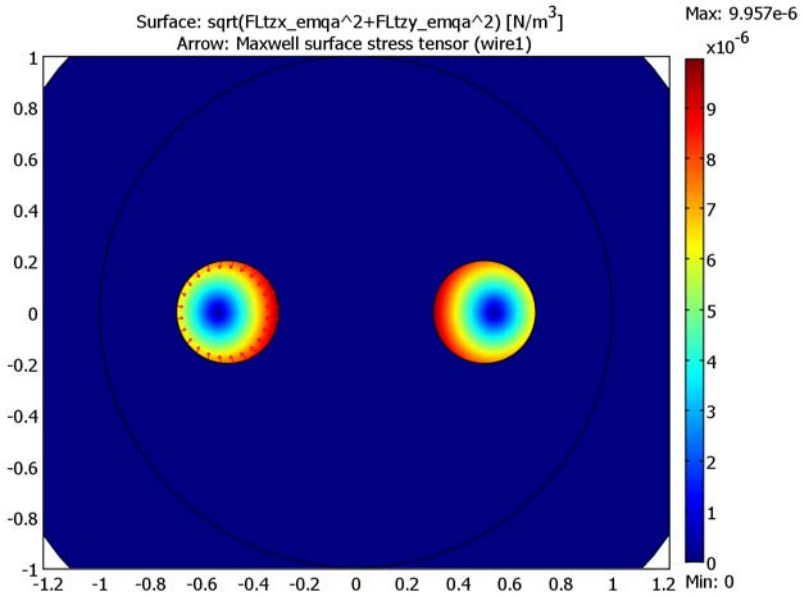


Figure 4: Stress tensor and magnetic field.

COMPUTING THE ELECTROMAGNETIC FORCE

Finally, compute the resulting forces on the wires. You can do this in three different ways.

First, compute the resulting force in the x direction on the left wire using the stress tensor method as follows:

1 Choose **Postprocessing>Data Display>Global** to open the **Global Data Display** dialog box.

2 In the **Expression** edit field type `wire1_forcex_emqa`, then click **OK**.

You should get a result near -1.99^{-7} N in the message log. The result here is given in N, while the results below are given in N/m. This is due to a difference in the integration between the force variable calculated from the stress tensor and the

integration in the postprocessing routine. The force variable assumes that the 2D domain is 1 m thick and multiply all integrations with this thickness. Note that the value of the force is the same, but it is either interpreted as a force on a 1 m section or as a force per length.

To compute the resulting force by integrating $\mathbf{J} \times \mathbf{B}$, perform the following steps.

1. Choose **Postprocessing>Subdomain Integration**.
2. Select Subdomain 3 and type `-Jez_emqa*By_emqa` in the **Expression** edit field inside the **Expression to integrate** area.
3. Finally, click **OK** to evaluate the integral.

   The result displayed in the message log is again approximately $-1.99 \cdot 10^{-7}$ N/m. The force in the *y* direction should be negligible. Confirm this by integrating `Jez_emqa*Bx_emqa` on Subdomain 3.

To compute the electromagnetic force using the principle of virtual work, you use the results from the geometric parameterization and sensitivity analysis. The forces computed in the previous section were obtained from the ordinary PDE solution, also known as the forward solution. The adjoint sensitivity solver also returns a second solution, which gives you the derivative of the total energy with respect to the displacement variables, `deltaX` and `deltaY`. Only the `deltaX` variable is of interest in this model because there is no net force in the *y* direction.

1. Choose **Postprocessing>Data Display>Global**.
2. In the **Expression** edit field, type `fsens(deltaX)`.
3. Click **OK**.

   Again, you obtain a result near $-2 \cdot 10^{-7}$ (N) in the message log.

By refining the mesh and re-solving the problem, you can verify that the solution indeed converges to $-2 \cdot 10^{-7}$ N/m.