COMSOL MULTIPHYSICS

AC/DC MODULE

MODEL LIBRARY

VERSION 3.4

COMSOL

How to contact COMSOL:

Benelux
COMSOL BV
Röntgenlaan 19
2719 DX Zoetermeer
The Netherlands
Phone: +31 (0) 79 363 4230
Fax: +31 (0) 79 361 4212
info@femlab.nl
www.femlab.nl

Denmark
COMSOL A/S
Diplomvej 376
2800 Kgs. Lyngby
Phone: +45 88 70 82 00
Fax: +45 88 70 80 90
info@comsol.dk
www.comsol.dk

Finland
COMSOL OY
Arabianranta 6
FIN-00560 Helsinki
Phone: +358 9 2510 400
Fax: +358 9 2510 4010
info@comsol.fi
www.comsol.fi

France
COMSOL France
WTC, 5 pl. Robert Schuman
F-38000 Grenoble
Phone: +33 (0)4 76 46 49 01
Fax: +33 (0)4 76 46 07 42
info@comsol.fr
www.comsol.fr

Germany
FEMLAB GmbH
Berliner Str. 4
D-37073 Göttingen
Phone: +49-551-99721-0
Fax: +49-551-99721-29
info@femlab.de
www.femlab.de

Italy
COMSOL S.r.l.
Via Vittorio Emanuele II, 22
25122 Brescia
Phone: +39-030-3793800
Fax: +39-030-3793899
info.it@comsol.com
www.it.comsol.com

Norway
COMSOL AS
Søndre gate 7
NO-7485 Trondheim
Phone: +47 73 84 24 00
Fax: +47 73 84 24 01
info@comsol.no
www.comsol.no

Sweden
COMSOL AB
Tegnérgatan 23
SE-111 40 Stockholm
Phone: +46 8 412 95 00
Fax: +46 8 412 95 10
info@comsol.se
www.comsol.se

Switzerland
FEMLAB GmbH
Technoparkstrasse 1
CH-8005 Zürich
Phone: +41 (0)44 445 2140
Fax: +41 (0)44 445 2141
info@femlab.ch
www.femlab.ch

United Kingdom
COMSOL Ltd.
UH Innovation Centre
College Lane
Hatfield
Hertfordshire AL10 9AB
Phone:+44-(0)-1707 284747
Fax: +44-(0)-1707 284746
info.uk@comsol.com
www.uk.comsol.com

United States
COMSOL, Inc.
1 New England Executive Park
Suite 350
Burlington, MA 01803
Phone: +1-781-273-3322
Fax: +1-781-273-6603

COMSOL, Inc.
10850 Wilshire Boulevard
Suite 800
Los Angeles, CA 90024
Phone: +1-310-441-4800
Fax: +1-310-441-0868

COMSOL, Inc.
744 Cowper Street
Palo Alto, CA 94301
Phone: +1-650-324-9935
Fax: +1-650-324-9936

info@comsol.com
www.comsol.com

For a complete list of international representatives, visit
www.comsol.com/contact

Company home page
www.comsol.com

COMSOL user forums
www.comsol.com/support/forums

AC/DC Module Model Library

Version: October 2007 COMSOL 3.4

CONTENTS

Chapter 1: Introduction

Chapter 2: Tutorial Models

Chapter 3: Motors and Drives Models

1

Introduction

The *AC/DC Module Model Library* consists of a set of models from various areas of low-frequency electromagnetic field simulations. Their purpose is to assist you in learning, by example, how to model sophisticated electromagnetic components, systems, and effects. Through them, you can tap the expertise of the top researchers in the field, examining how they approach some of the most difficult modeling problems you might encounter. You can thus get a feel for the power that COMSOL Multiphysics® offers as a modeling tool. In addition to serving as a reference, the models can also give you a big head start if you are developing a model of a similar nature.

We have divided these models into four groups: electrical components, general industrial applications, motor and drives, and tutorial models. The models also illustrate the use of the various electromagnetics-specific application modes from which we built them. These specialized modes are unavailable in the base COMSOL Multiphysics package, and they come with their own graphical-user interfaces that make it quick and easy to access their power. You can even modify them for custom requirements. COMSOL Multiphysics itself is very powerful and, with sufficient expertise in a given field, you certainly could develop these modes by yourself—but why spend the hundreds or thousands of hours that would be necessary when our team of experts has already done the work for you?

Note that the model descriptions in this book do not contain every detail on how to carry out every step in the modeling process. Before tackling these in-depth models, we urge you to first read the second book in the AC/DC Module documentation set. Titled the *AC/DC Module User's Guide*, it introduces you to the basic functionality in the module, reviews new features in the version 3.4 release, covers basic modeling techniques and includes reference material of interest to those working in electromagnetics. A third book, the *AC/DC Module Reference Guide*, contains reference material about application mode implementations and command-line programming. It is available in HTML and PDF format from the COMSOL Help Desk. For more information on how to work with the COMSOL Multiphysics graphical user interface, please refer to the *COMSOL Multiphysics User's Guide* or the *COMSOL Multiphysics Quick Start* manual. Extensive information about command-line modeling using COMSOL Script is available in yet another book, the *COMSOL Multiphysics Scripting Guide*.

The book in your hands, the *AC/DC Module Model Library*, provides details about a large number of ready-to-run models that illustrate real-world uses of the module. Each entry comes with theoretical background as well as instructions that illustrate how to set it up. They were written by our staff engineers who have years of experience in electromagnetics; they are your peers, using the language and terminology needed to get across the sophisticated concepts in these advanced topics.

Finally note that we supply these models as COMSOL Multiphysics Model MPH-files so you can import them into COMSOL Multiphysics for immediate execution, allowing you to follow along with these examples every step along the way.

Model Library Guide

The table below summarizes key information about the entries in the AC/DC Module Model Library. A series of columns states the application mode (such as Perpendicular Currents) used to solve the corresponding model. The solution time is the elapsed time measured on a machine running Windows Vista with a 2.6 GHz AMD Athlon X2 Dual Core 500 CPU and 2 GB of RAM. For models with a sequential solution strategy, the Solution Time column shows the elapsed time for the longest solution step. Additional columns point out the solution properties a given example highlights.

The choices here include the type of analysis (such as time dependent) and whether multiphysics or parametric studies are included.

TABLE 1-1: AC/DC MODULE MODEL LIBRARY

MODEL	PAGE	APPLICATION MODE	SOLUTION TIME	STATIONARY	TIME-HARMONIC	TIME DEPENDENT	EIGENFREQUENCY/EIGENMODE	NONLINEAR	MULTIPHYSICS	PARAMETRIC STUDY
TUTORIAL MODELS										
Eddy Currents	21*	Azimuthal Induction Currents	1 s		√					√
Floating Potentials and Electric Shielding	33*	Conductive Media DC	11 s	√						
Coil with Infinite Elements	46*	Azimuthal Induction Currents	3 s	√						
Microstrip	66*	Electrostatics, In-Plane Electric and Induction Currents, Potentials	5 s	√	√					
Small-Signal Analysis of an Inductor	79*	Azimuthal Induction Currents	5 s	√	√					
Electromagnetic Forces on Parallel Current-Carrying Wires	8	Perpendicular Currents	1 s	√						
Inductive Heating of a Copper Cylinder	16	Azimuthal Currents, Heat Transfer	9 s		√	√		√	√	
Permanent Magnet	21	Magnetostatics, No Currents	26 s	√						
MOTORS AND DRIVES MODELS										

TABLE 1-1: AC/DC MODULE MODEL LIBRARY

MODEL	PAGE	APPLICATION MODE	SOLUTION TIME	STATIONARY	TIME-HARMONIC	TIME DEPENDENT	EIGENFREQUENCY/EIGENMODE	NONLINEAR	MULTIPHYSICS	PARAMETRIC STUDY
Linear Electric Motor of the Moving Coil Type	30	Perpendicular Currents	2 s	√						
Generator in 2D	39	Perpendicular Currents, Moving Mesh	2 min			√		√	√	
Generator with Mechanical Dynamics and Symmetry	53	Perpendicular Currents	4 min			√		√	√	
Generator in 3D	65	Magnetostatics, No Currents	6 min	√				√		
Magnetic Brake in 3D	72	3D Magnetostatics	10 min	√						
Railgun	84	3D Quasi-Statics, Conductive Media DC	43 min			√			√	
ELECTRICAL COMPONENT MODELS										
Tunable MEMS Capacitor	100	Electrostatics	8 s	√						
Induction Currents from Circular Coils	110	Azimuthal Currents	10 s		√	√				
Magnetic Field of a Helmholtz Coil	122	3D Magnetostatics	15 s	√						
Inductance in a Coil	131	Azimuthal Currents	12 s		√					√
Integrated Square-Shaped Spiral Inductor	141	3D Magnetostatics	27 s	√						
Inductance of a Power Inductor	153	3D Quasi-Statics	13 min		√					
Bonding Wires to a Chip	161	3D Quasi-Statics	¤17 min		√					√

TABLE 1-1: AC/DC MODULE MODEL LIBRARY

MODEL	**PAGE**	**APPLICATION MODE**	**SOLUTION TIME**	**STATIONARY**	**TIME-HARMONIC**	**TIME DEPENDENT**	**EIGENFREQUENCY/EIGENMODE**	**NONLINEAR**	**MULTIPHYSICS**	**PARAMETRIC STUDY**
High Current Cables in a Circuit	170	Joule Heating	3 min	√					√	
Inductor in Amplifier Circuit	189	Azimuthal Currents, ODE	2 min	√		√		√	√	√
GENERAL INDUSTRIAL APPLICATION MODELS										
Eddy Currents in 3D	202	3D Quasi-Statics	6 min		√					
Cold Crucible	211	3D Quasi-Statics	20 s		√					
Electric Impedance Sensor	222	Small In-Plane Currents	2 min		√					√
One-Sided Magnet and Plate	229	Magnetostatics, No Currents	41 s	√						
Magnetic Signature of a Submarine	241	Magnetostatics, No Currents	9 s	√						
3D Quadrupole Lens	252	Magnetostatics, No Currents	20 s	√						
Superconducting Wire	259	PDE, General Form	56 s			√		√		
RF-Heated Hot Wall Furnace for Semiconductor Processing	267	3D Quasi-Statics, General Heat Transfer	4 min		√	√			√	
An RFID System	292	Magnetostatics	7 min	√						

* This page number refers to the *AC/DC Module User's Guide*.

¤ This model requires a 64-bit platform.

Typographical Conventions

All COMSOL manuals use a set of consistent typographical conventions that should make it easy for you to follow the discussion, realize what you can expect to see on the screen, and know which data you must enter into various data-entry fields. In particular, you should be aware of these conventions:

- A **boldface** font of the shown size and style indicates that the given word(s) appear exactly that way on the COMSOL graphical user interface (for toolbar buttons in the corresponding tooltip). For instance, we often refer to the **Model Navigator**, which is the window that appears when you start a new modeling session in COMSOL; the corresponding window on the screen has the title **Model Navigator**. As another example, the instructions might say to click the **Multiphysics** button, and the boldface font indicates that you can expect to see a button with that exact label on the COMSOL user interface.
- The names of other items on the graphical user interface that do not have direct labels contain a leading uppercase letter. For instance, we often refer to the Draw toolbar; this vertical bar containing many icons appears on the left side of the user interface during geometry modeling. However, nowhere on the screen will you see the term “Draw” referring to this toolbar (if it were on the screen, we would print it in this manual as the **Draw** menu).
- The symbol **>** indicates a menu item or an item in a folder in the **Model Navigator**. For example, **Physics>Equation System>Subdomain Settings** is equivalent to: On the **Physics** menu, point to **Equation System** and then click **Subdomain Settings**. **COMSOL Multiphysics>Heat Transfer>Conduction** means: Open the **COMSOL Multiphysics** folder, open the **Heat Transfer** folder, and select **Conduction**.
- A `Code` (monospace) font indicates keyboard entries in the user interface. You might see an instruction such as “Type `1.25` in the **Current density** edit field.” The monospace font also indicates COMSOL Script codes.
- An *italic* font indicates the introduction of important terminology. Expect to find an explanation in the same paragraph or in the Glossary. The names of books in the COMSOL documentation set also appear using an italic font.

2

Tutorial Models

This chapter contains a selection of models that demonstrate important features and modeling techniques. Therefore, this chapter contains models from many different application areas. For pedagogical reasons, these models are kept as simple as possible with the main focus on the feature that they demonstrate.

Electromagnetic Forces on Parallel Current-Carrying Wires

Introduction

One ampère 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 do not have a negligible cross section. However, by successively shrinking the radius of the wires, the force would approach $2 \cdot 10^{-7}$ N/m.

The force between the wires is computed using two different methods: by integrating the volume force density over the wire cross section and by integrating the stress tensor on the boundary. The results are in good agreement and are close but not equal to the number $2 \cdot 10^{-7}$ N for the 1 ampère 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 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

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

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 $-1.92\cdot10^{-7}$ N. 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 $-1.92\cdot10^{-7}$ N. The number of mesh elements and the finite size of the geometry is what ultimately limits the accuracy in the calculations.

Model Library path: `ACDC_Module/Tutorial_Models/parallel_wires`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **2D** in the **Space dimension** list.

2 Select the **AC/DC Module>Statics>Magnetostatics>Perpendicular Induction Currents, Vector Potential** application mode.

3 Click **OK**.

OPTIONS AND SETTINGS

1 Choose **Options>Constants**, then enter the following names, expressions, and (optionally) descriptions; when finished, click **OK**.

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 $J_0 = I_0/A$.

2 Choose **Options>Axes/Grid Settings**, then specify axis limits and grid settings according to the following table; when finished, click **OK**.

AXIS		GRID	
x min	-6	x spacing	0.5
x max	6	Extra x	-0.3 0.7
y min	-6	y spacing	0.5
y max	6	Extra y	0.2

As you can see on your computer screen, the aspect ratio of the drawing area is preserved. The specified min/max values apply to either the width or the height in such a way that the whole of the region in the *xy*-plane that you specify is visible. The latter rule still applies if the aspect ratio changes as a result of modified settings for the Model Tree. If you modify the size of the GUI window, however, you lose the settings for the axis limits.

GEOMETRY MODELING

1 Draw a circle C1 with radius 0.2 centered at (–0.5, 0).

2 Draw a circle C2 with radius 0.2 centered at (0.5, 0).

3 Draw a circle C3 with radius 5 centered at (0, 0).

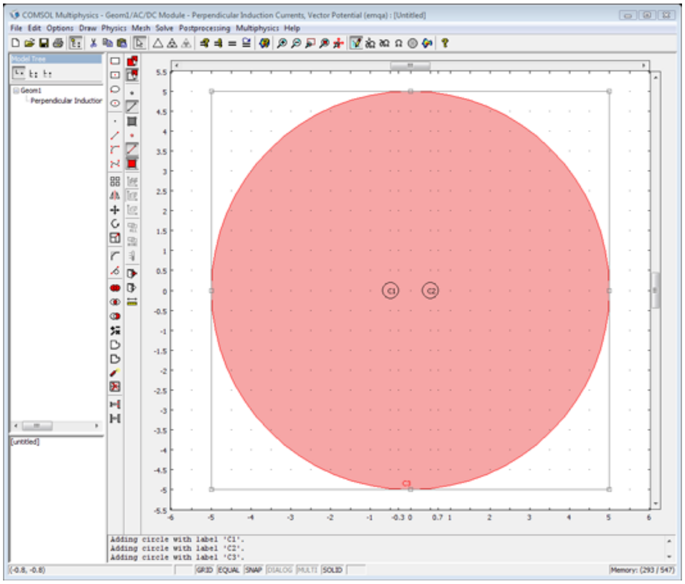


Figure 2-1: The model geometry.

PHYSICS SETTINGS

Boundary Conditions

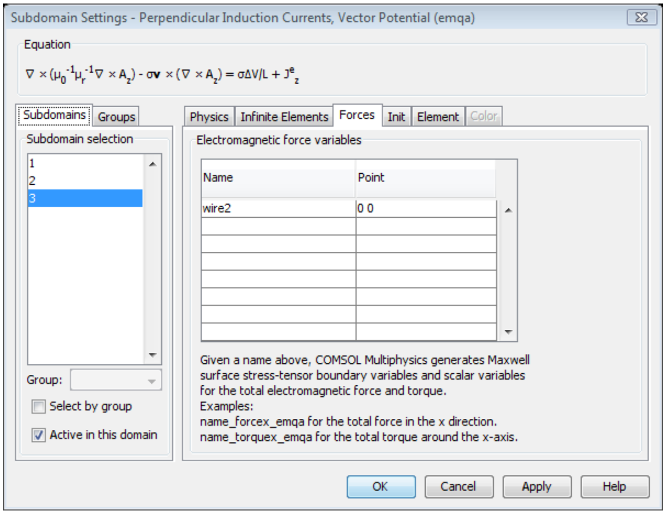
The default boundary conditions are used.

Subdomain Settings

Apply an **External current density** and define **Electromagnetic force variables** according to the following table.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAIN 3
J_z^e	`0`	`J0`	`-J0`
Electromagnetic force variables		`wire1`	`wire2`

Leave the other parameters at their default values.



MESH GENERATION

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

2 Click the **Refine Mesh** button to refine the mesh once.

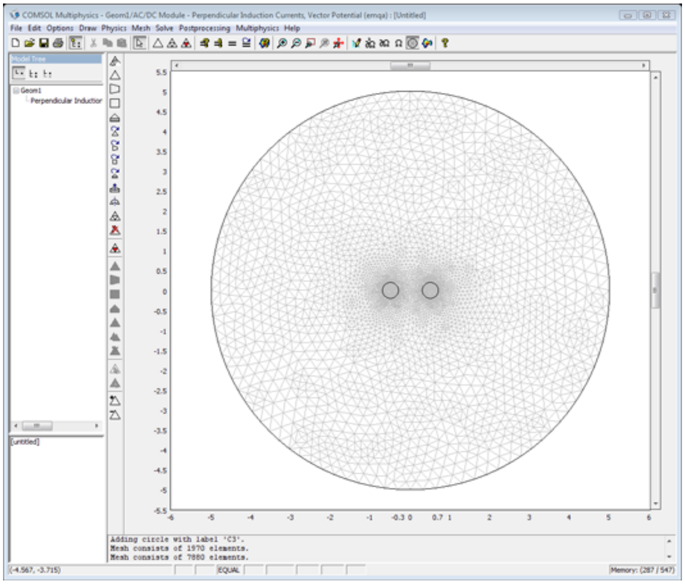


Figure 2-2: The mesh.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows the magnetic flux density. 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(Jez_emqa^2*(Bx_emqa^2+By_emqa^2))`.

3 Click the **Contour** tab. Select the **Contour plot** check box and select **Magnetic field, norm** from the **Predefined quantities** list.

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

5 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 **OK**.

Figure 2-3 shows the result. The volume force density is unevenly distributed within each wire due to the slightly inhomogeneous magnetic field from the other wire. If you decrease the radii of the wires, the force density becomes evenly distributed. In the definition of 1 A it is assumed that the radii are negligible. The field becomes homogeneous as the radii approach zero. The effects of the self-induced field of a wire then cancel out.

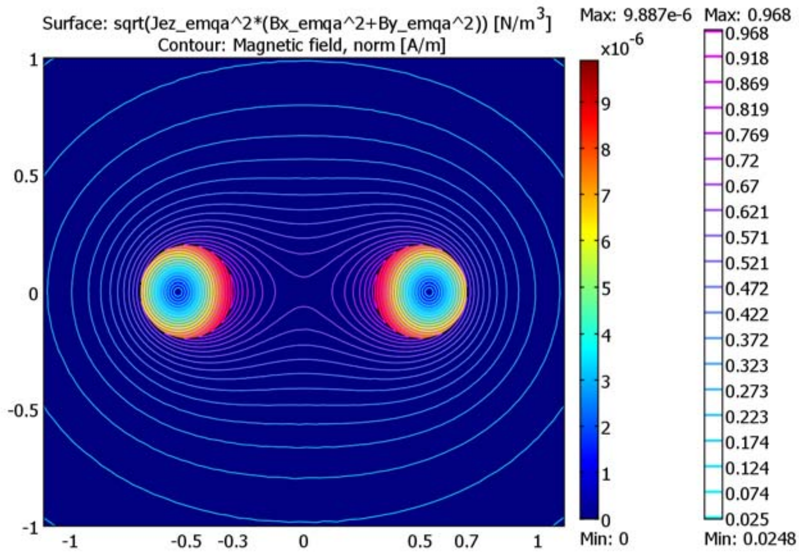


Figure 2-3: Force density and magnetic field.

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

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

7 Click the **Arrow** tab. Select the **Arrow plot** check box and select **Boundaries** from the **Plot arrows on** list. On the **Boundary Data** page select **Maxwell surface stress tensor (wire1)** from the **Predefined quantities** list.

8 In the **Arrow parameters** area clear the **Scale factor: Auto** check box and type `0.1` in the associated edit field.

9 Click **OK** to close the dialog box and generate the plot shown in Figure 2-4.

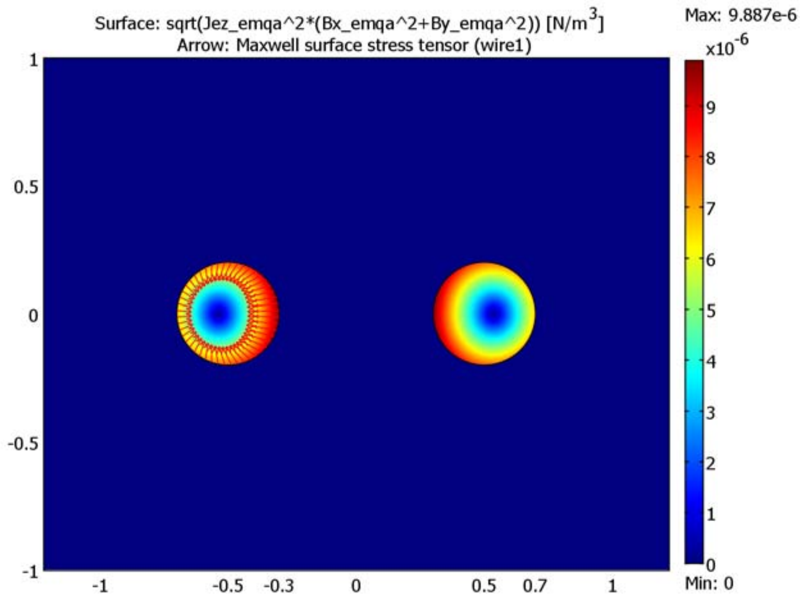


Figure 2-4: Stress tensor and magnetic field.

Finally, compute the resulting forces on the wires.

1 To compute the resulting force in the x direction on the left wire using the stress tensor method, choose **Postprocessing>Data Display>Global** to open the **Global Data Display** dialog box. In the **Expression to evaluate** edit field type `wire1_forcex_emqa`, then click **OK**.

   You should get a result near $-1.92 \cdot 10^{-7}$ (N/m) in the message log.

2 To alternatively compute the resulting force by integrating $\mathbf{J} \times \mathbf{B}$, choose **Postprocessing>Subdomain Integration**. Select Subdomain 2 and type `-Jez_emqa*By_emqa` in the **Expression** edit field inside the **Expression to integrate** area. Finally, click **OK** to evaluate the integral.

   The result displayed in the message log is again approximately $-1.92 \cdot 10^{-7}$ N/m. The force in the y direction should be negligible. Confirm this by integrating `Jez_emqa*Bx_emqa` on Subdomain 2.

Inductive Heating of a Copper Cylinder

Introduction

The induced currents in a copper cylinder produce heat, and when the temperature rises, the electric conductivity of the copper changes. Solving the heat transfer simultaneously with the field propagation is therefore crucial for an accurate description of this process.

The heating caused by the induced currents is called inductive heating. Generally heating due to currents is also called resistive heating or ohmic heating.

Model Definition

The system to be solved is given by

$$j\omega\sigma(T)\mathbf{A} + \nabla \times (\mu^{-1}\nabla \times \mathbf{A}) = \mathbf{0}$$

$$\rho C_p \frac{\partial T}{\partial t} - \nabla \cdot k\nabla T = Q(T, \mathbf{A})$$

where ρ is the density, C_p is the specific heat capacity, k is the thermal conductivity, and Q is the inductive heating.

The electric conductivity of copper, σ, is given by the expression

$$\sigma = \frac{1}{[\rho_0(1 + \alpha(T - T_0))]}$$

where ρ_0 is the resistivity at the reference temperature $T_0 = 293$ K, α is the temperature coefficient of the resistivity, and T is the actual temperature in the domain.

The time average of the inductive heating over one period, is given by

$$Q = \frac{1}{2}\sigma|\mathbf{E}|^2$$

Results and Discussion

The temperature after 1200 s is shown in Figure 2-5 below. The average temperature of the copper cylinder has increased from 293 K to 300 K during this time. The current in the coil has an amplitude of 1 kA.

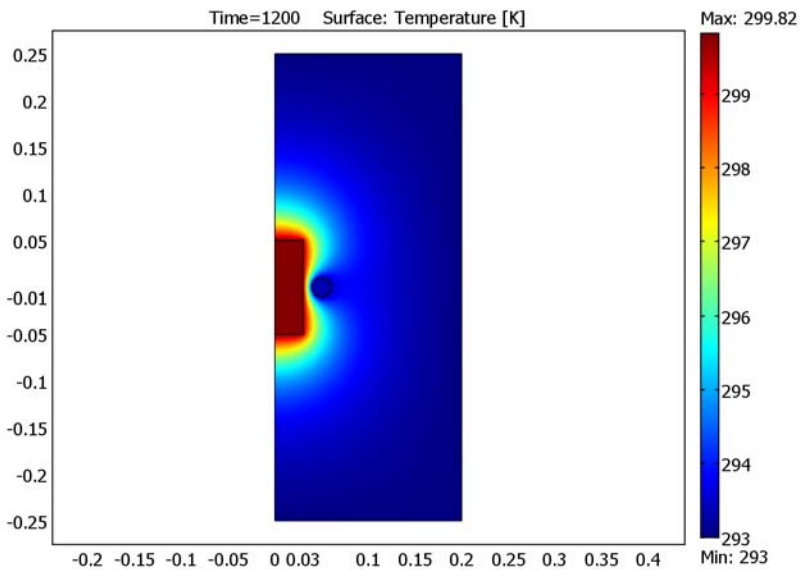


Figure 2-5: Temperature distribution in the copper cylinder and its surroundings after 1200 s.

Model Library path: `ACDC_Module/Tutorial_Models/inductive_heating`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Select the **Axial symmetry (2D)** in the **Space dimension** list.
2. Select the **AC/DC Module>Electro-Thermal Interaction>Azimuthal Induction Heating>Transient analysis** multiphysics mode.
3. Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu choose **Axes/Grid Settings**.

2 Specify axis and grid settings according to the following table.

AXIS	VALUE	GRID	VALUE
r min	-0.05	r spacing	0.05
r max	0.5	Extra r	0.03
z min	-0.3	z spacing	0.05
z max	0.3	Extra z	-0.01 0.01

3 From the **Options** menu choose **Constants**.

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

NAME	EXPRESSION	DESCRIPTION
I0	1e3[A]	Current
diam	0.02[m]	Diameter
circ	pi*diam	Circumference
Js0	I0/circ	Surface current density
T0	293[K]	Reference temperature
r0	1.754e-8[ohm*m]	Resistivity at T = T0
alpha	0.0039[1/K]	Temperature coefficient
rho1	1.293[kg/m^3]	Density of air
Cp1	1.01e3[J/(kg*K)]	Heat capacity of air
k1	0.026[W/(m*K)]	Thermal conductivity of air
rho2	8930[kg/m^3]	Density of copper
Cp2	340[J/(kg*K)]	Heat capacity of copper
k2	384[W/(m*K)]	Thermal conductivity of copper

GEOMETRY MODELING

1 Draw a circle C1 with radius 0.01 centered at (0.05, 0).

2 Draw a rectangle R1 with opposite corners at (0, –0.25) and (0.2, 0.25).

3 Draw a rectangle R2 with opposite corners at (0, –0.05) and (0.03, 0.05).

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

PHYSICS SETTINGS

Application Scalar Variables

In the **Application Scalar Variables** dialog box set the frequency nu_qa to 500 Hz.

Boundary Conditions

1 Select the **Azimuthal Currents** application mode from the **Multiphysics** menu.

2 Open the **Boundary Settings** dialog box, and select the **Interior boundaries** check box.

3 Enter boundary coefficients according to the following table.

SETTINGS	BOUNDARIES 1, 3, 5	BOUNDARIES 2, 7, 9	BOUNDARIES 10-13
Boundary condition	Axial symmetry	Magnetic insulation	Surface current
$\mathbf{J}_{s\varphi}$			Js0

4 Select the **Heat Transfer** application mode from the **Multiphysics** menu.

5 Enter boundary coefficients according to the following table.

SETTINGS	BOUNDARIES 1, 3, 5	BOUNDARIES 2, 7, 9
Boundary condition	Axial symmetry	Temperature
T_0		T0

Subdomain Settings

1 Select the **Azimuthal Currents** application mode from the **Multiphysics** menu.

2 Enter the electric conductivity according to the following table.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAIN 3
σ	0	1/(r0*(1+ alpha*(T-T0)))	0

3 Select the **Heat Transfer** application mode from the **Multiphysics** menu.

4 Enter the expressions according to the following table.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAIN 3
k	k1	k2	k2
ρ	rho1	rho2	rho2
C_p	Cp1	Cp2	Cp2

The variable Qav_qa is the resistive heating defined by the Azimuthal Currents application mode. This variable is automatically inserted as the heat source in the Heat Transfer application mode.

Initial Conditions

On the **Init** tab select all domains in the **Domain selection** list and set the initial conditions to `T0`.

MESH GENERATION

Click the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

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

2 In the **Times** edit field on the **General** page type `linspace(0,1200,21)`.

3 On the **Advanced** page, select the **Use complex functions with real input** check box.

4 Click **OK** to close the dialog box.

5 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot should closely resemble that in Figure 2-5 on page 17, which visualizes the temperature distribution at the end of the simulation period.

To view the temperature distribution as a function of time, click the **Animate** button on the Plot toolbar.

Permanent Magnet

Introduction

In magnetostatic problems, where no currents are present, the problem can be solved using a scalar magnetic potential. This model illustrates this technique for a horse shoe-shaped permanent magnet placed near an iron rod.

Figure 2-6: A full 3D view of the geometry. Left-right and top-down symmetry is used to minimize the problem size.

Model Definition

In a current free region, where

$$\nabla \times \mathbf{H} = \mathbf{0}$$

it is possible to define the scalar magnetic potential, V_m, from the relation

$$\mathbf{H} = -\nabla V_m$$

This is analogous to the definition of the electric potential for static electric fields.

Using the constitutive relation between the magnetic flux density and magnetic field

$$\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$$

together with the equation

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

you can derive an equation for V_{m},

$$-\nabla \cdot (\mu_0 \nabla V_{\mathrm{m}} - \mu_0 \mathbf{M}_0) = 0$$

Results and Discussion

The force on the rod is calculated by integrating the surface stress tensor over all boundaries of the rod. The expression for the stress tensor reads,

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

where $\mathbf{n}_1$ is the boundary normal pointing out from the rod and T_2 the stress tensor of air. The integration gives 148 N, which corresponds to one quarter of the rod. The actual force on the rod is therefore four times this value, 592 N.

Model Library path: `ACDC_Module/Tutorial_Models/rod_and_permanent_magnet`

Modeling Using the Graphical User Interface

Select the **3D>AC/DC Module>Statics>Magnetostatics, No Currents** mode in the **Model Navigator**.

OPTIONS AND SETTINGS

In the **Constants** dialog box enter the following names and expressions.

NAME	EXPRESSION	DESCRIPTION
murFe	5000	Relative permeability of iron
Mpre	750000	Magnetization of magnet (A/m)

GEOMETRY MODELING

You can use symmetries to reduce the size of the model. Only one arm of the magnet needs to be included in the model, and a horizontal cut can also be made. Thus, only one quarter of the magnet and rod will be modeled.

1 Create a work plane by opening the **Work-Plane Settings** dialog box and clicking **OK** to obtain the default work plane in the *xy*-plane.

2 In the **Axes/Grid Settings** dialog box, clear the **Auto** check box on the **Grid** tab and set both the grid **x spacing** and **y spacing** to `0.2`.

3 Draw a rectangle R1 with corners at (0, 0.2) and (0.4, 0.4).

4 Draw another rectangle R2 with corners at (0.4, 0.2) and (0.6, 0.4).

5 Click the **Ellipse/Circle (Centered)** button on the Draw toolbar. Draw a circle C1 with center at (0.6, 0) and radius 0.2. Use the right mouse button to make sure a circle is obtained rather than an ellipse.

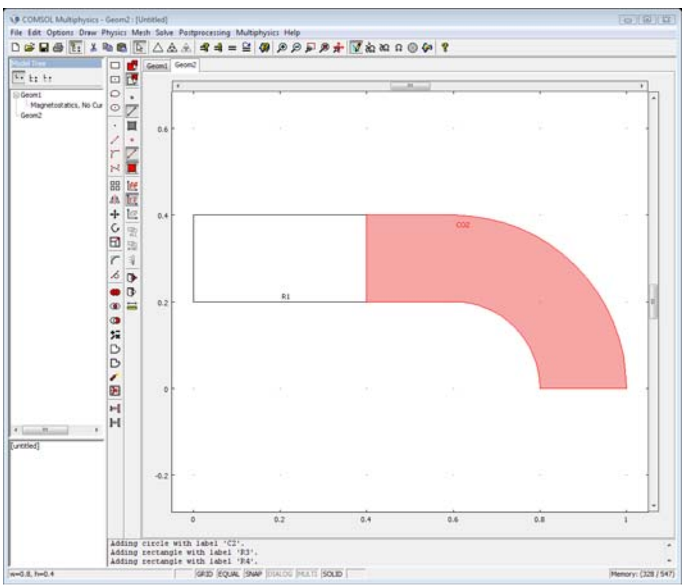
6 Draw another circle C2 also with center at (0.6, 0) but with radius 0.4.

7 Select both circles and click the **Difference** button to create the object CO1.

8 Draw a rectangle R3 with corners at (0.2, 0) and (0.6, 0.4), and a rectangle R4 with corners at (0.2, –0.4) and (1, 0).

9 Select CO1, R3, and R4 and click the **Difference** button to create the object CO2.

10 Select CO2 and R2 and click the **Union** button. Then click the **Delete Interior Boundaries** button.



11 Open the **Extrude** dialog box and select both R1 and CO2. Enter the **Distance** `0.1` and click **OK** to extrude the magnet.

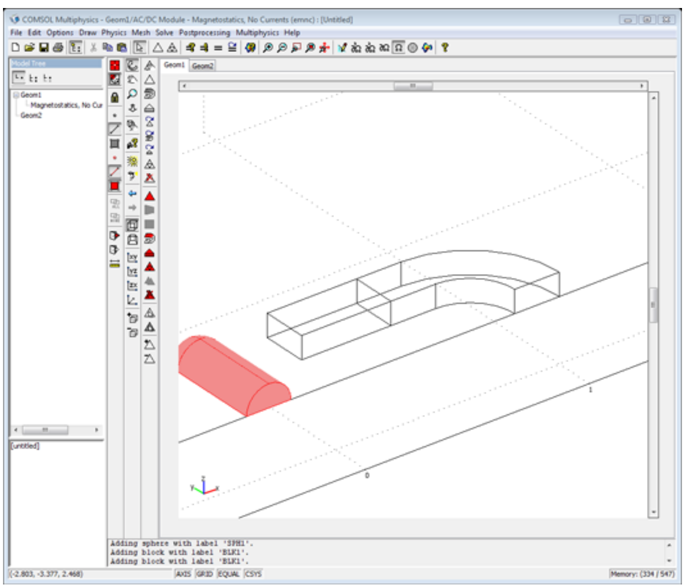
12 Create the rod by combining a cylinder and a sphere. Click the **Cylinder** button to open the **Cylinder** dialog box, and enter the **Radius** `0.1` and **Height** `0.4`. Set the **Axis base point** to (–0.3, 0, 0) and the **Axis direction vector** to (0, 1, 0). Click **OK** to create the cylinder.

13 Click the **Sphere** button and enter the **Radius** `0.1`. Set the **Axis base point** to (–0.3, 0.4, 0) then click **OK** to create the sphere.

14 Half of the rod should now be removed. This can be done by subtracting a block from the cylinder and sphere. Click the **Block** button and enter the **Length** coordinates (0.4, 0.6, 0.2). Set the **Axis base point** to (–0.5, –0.1, –0.2), then click **OK** to create a block.

15 Open the **Create Composite Object** dialog box, and type the formula `CYL1+SPH1-BLK1` in the **Set formula** edit field. Clear the **Keep interior boundaries** check box and then click **OK**.

16 Finally add a block surrounding the magnets and rod. In the **Block** dialog box use the **Length** coordinates (5, 2, 2) and the **Axis base point** (–2, 0, 0).



PHYSICS SETTINGS

Boundary Conditions

Along the boundaries far away from the magnet, the magnetic field should be tangential to the boundary as the flow lines should form closed loops around the magnet. The natural boundary condition is

$$\mathbf{n} \cdot (\mu_0 \nabla V_{\mathrm{m}} - \mu_0 \mathbf{M}_0) = \mathbf{n} \cdot \mathbf{B} = 0$$

Thus the magnetic field is made tangential to the boundary by a Neumann condition on the potential.

Along the symmetry boundary below the magnet, the magnetic field should also be tangential, and therefore you can apply the same Neumann condition there.

On the other hand, the magnetic field should be normal to the other symmetry boundary at the front in the figure above in order for the magnetic flow lines to form closed loops around the magnet. This means that the magnetic field is symmetric with respect to the boundary. This can be achieved by setting the potential to zero along the boundary, and thus making the potential antisymmetric with respect to the boundary.

Enter these boundary conditions according to the following table.

SETTINGS	BOUNDARIES 2, 8, 24	ALL OTHERS
Boundary condition	Zero potential	Magnetic insulation

Subdomain Settings

In the **Subdomain Settings** dialog box, enter the coefficients according to the table below.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAIN 3	SUBDOMAIN 4
Constitutive relation	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$	$\mathbf{B} = \mu_0\mathbf{H} + \mu_0\mathbf{M}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$
μ_r	1	`murFe`		`murFe`
$\mathbf{M}$			`Mpre 0 0`	
Force variables		`rod`		

MESH GENERATION

1 Choose **Mesh>Free Mesh Parameters**.

2 From the **Predefined mesh sizes** list select **Fine**.

3 In order to make the mesh finer in the magnet and in the rod, click the **Subdomain** tab, then set the **Maximum element size** for Subdomains 2, 3, and 4 to `0.025`.

4 Click **OK** to close the dialog box.

5 Click the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows a slice plot of the magnetic potential.

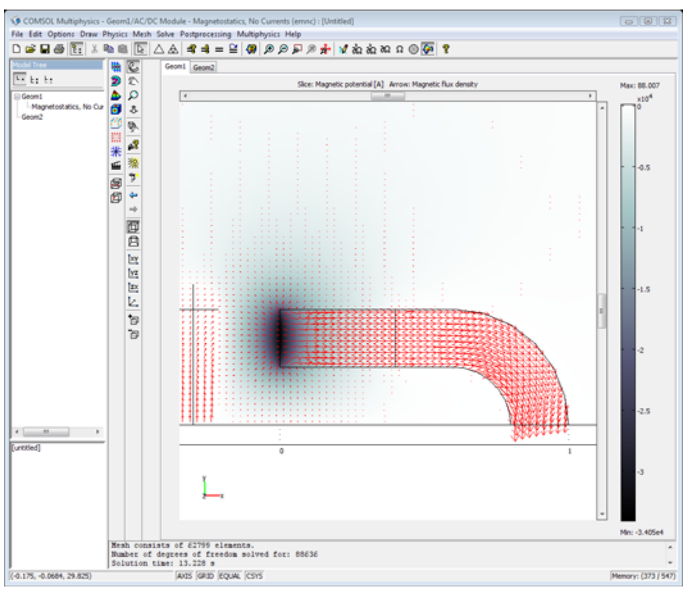
1 In the **Plot Parameters** dialog box, select **Slice** and **Arrow** plot.

2 On the **Slice** tab set the **x levels** to `0` and the **y levels** to `0` in the **Slice positioning** area. For the **z levels** select **Vector with coordinates** and enter `0.05`. Select **bone** from the **Colormap** list.

3 Click the **Arrow** tab, select the **Magnetic flux density** as **Arrow data**. Set the **x points** to `200` and the **y points** to `100` in the **Arrow positioning** area. For the **z points** select **Vector with coordinates** and enter `0.051`. Select **Arrow** from the **Arrow type** list.

4 Click **OK** to close the dialog and generate the plot.

5 Click the button **Goto XY View** and use the zoom tool to zoom in on the magnet and rod.



Force Calculation

To calculate the force on the rod, use the surface stress tensor, which is derived in "Electromagnetic Forces" on page 110 in the *AC/DC Module User's Guide*,

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

where $\mathbf{n}_1$ is the boundary normal pointing out from the rod and T_2 the stress tensor of air. The electromagnetic force variable `rod`, which was defined on the rod in the **Subdomain Settings** dialog box, generates the variables `rod_forcex_emnc`, `rod_forcey_emnc`, and `rod_forcez_emnc`, which are the forces on the rod in the x, y, and z directions, respectively.

To evaluate the force in the x direction, open the **Global Data Display** dialog box and enter the expression `rod_forcex_emnc`. The result is about 148 N. Because the model only includes one quarter of the rod, the actual force is four times this value, 592 N.

3

Motors and Drives Models

This chapter contains examples of models in the area of electric motors and drives. It includes static models with force computations as well as transient models with motion.

Linear Electric Motor of the Moving Coil Type[1]

Introduction

Linear electric motors (LEMs) are electromechanical devices that produce motion in a straight line, without the use of any mechanism to convert rotary motion to linear motion. The benefits over conventional rotary-to-linear counterparts are the absence of mechanical gears and transmission systems, which results in a higher dynamic performance and improved reliability. The acceleration available from these motors is remarkable compared to traditional motor drives that convert rotary to linear motion.

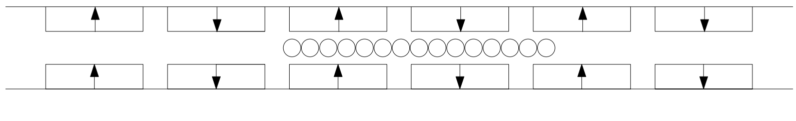
A LEM is similar to a rotary motor whose stator and rotor have been cut along a radial plane and unrolled so that it provides linear thrust. The same electromagnetic force that produces torque in a rotary motor produces a direct linear force in a linear motor.

The force that develops in the motor depends on the magnetic field and the current in the coils and of course on the shape of the different parts. A parameter of interest is the maximum force in standstill operation, that is, when the magnetic field is constant and the force depends only on the current density distribution. The current density in the coils is limited by the maximum temperature allowed in the coils. For a certain motor topology you can estimate the maximum electromagnetic force in standstill by first performing a heat transfer simulation to get the maximum current density, and then using this value in the magnetostatic problem where the forces are computed.

Model Definition

A LEM of moving coil type consists of a stationary primary and a moving secondary. The stationary primary is a permanent magnet assembly consisting of a double magnet array with altering polarities bounded inside a U-shaped iron former. The moving secondary is centered within the U-shaped magnet assembly, and consists of 3-phase core coils mounted on a T-shaped aluminum armature. The cross-sectional view of a moving coil is shown in the following figure.

1. Model provided by Hans Johansson and Mikael Hansson, The Royal Institute of Technology, Stockholm.



The permanent magnets in the primary have a residual induction B_r of 1.23 T, giving the magnetization

$$M = B_r/\mu_0 = 0.98 \cdot 10^6 \mathrm{A/m}$$

The current in the coil is obtained via a heat transfer simulation. If a temperature of 65 °C is allowed in the coil and the surrounding temperature is 25 °C, the simulation in the heat transfer mode gives a maximum heat source Q of 230,000 W/m^3.

The heat source is equivalent to the resistive losses in the copper, which are given by the expression

$$Q = \mathbf{J} \cdot \mathbf{E} = \frac{1}{\rho_{\mathrm{Cu}}} \cdot J_z^2$$

where ρ_{Cu} is the resistivity of copper, which is $1.72 \cdot 10^{-8}$ Ωm. This gives the maximum current density J_z of 3.66 A/mm^2.

EQUATION

COMSOL Multiphysics solves the problem using a magnetostatic equation for the magnetic potential **A**. This is a 2D model, so it is possible to formulate an equation for the z component of the potential,

$$-\nabla \cdot \left(\mu_0^{-1} \mu_r^{-1} \nabla A_z - \begin{bmatrix} -\mu_0^{-1} \mu_r^{-1} B_{ry} \\ \mu_0^{-1} \mu_r^{-1} B_{rx} \end{bmatrix} \right) = J_z^e$$

where μ_0 is the permeability of vacuum, μ_r the relative permeability, $\mathbf{B}_r$ the remanent flux density, and J_z^e is the current density.

Results and Discussion

The force on the coils can be calculated either using the method of virtual displacement making a small perturbation and calculating the change in magnetic energy

$$W_m = \frac{\mathbf{B} \cdot \mathbf{H}}{2}$$

or using the Lorentz force expression

$$\mathbf{F} = \mathbf{J} \times \mathbf{B}$$

The resulting force in the x direction is $F_x = 276$ N/m. The air gap shear stress τ is the force developed per unit air gap area and gives an indication of the specific force capability. $F/(l{\cdot}A)$ is the force per moving volume. This can be compared with the force per moving mass, which indicates the acceleration capabilities. The air gap length is 40 mm and the air gap area is 190 mm^2.

QUANTITY	RESULT
F/l	276 N/m
τ	0.69 N/cm^2
$F/(l{\cdot}A)$	1.45 N/cm^3

Model Library path: `ACDC_Module/Motors_and_Drives/coil_LEM`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **2D** in the Space dimension list.

2 Select the **AC/DC Module>Statics>Magnetostatics>Perpendicular Induction Currents, Vector Potential** application mode.

3 Select **Lagrange - Linear** as element type in the **Element** list.

4 Click **OK**.

OPTIONS AND SETTINGS

1 Set axis and grid settings according to the following table:

AXIS		GRID	
x min	`-0.06`	x spacing	`0.002`
x max	`0.06`	Extra x	

AXIS		GRID	
y min	-0.01	y spacing	0.01
y max	0.03	Extra y	0.0025 0.0035 0.0075 0.0105

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

NAME	EXPRESSION	DESCRIPTION
sigCu	5.99e7[S/m]	Conductivity of copper
sigFe	1.03e7[S/m]	Conductivity of iron
muFe	2e3	Relative permeability of iron
Br	1.23[T]	Remanent flux density of the magnets
Jin	3.66e6[A/m^2]	Current density

GEOMETRY MODELING

1 Draw a rectangle R1 with opposite corners at (–0.05, 0) and (0.05, 0.0105).

2 Draw a rectangle R2 with opposite corners at (–0.05, 0.0075) and (0.05, 0.0105).

3 Draw a rectangle R3 with opposite corners at (–0.058, 0.0035) and (–0.042, 0.0075).

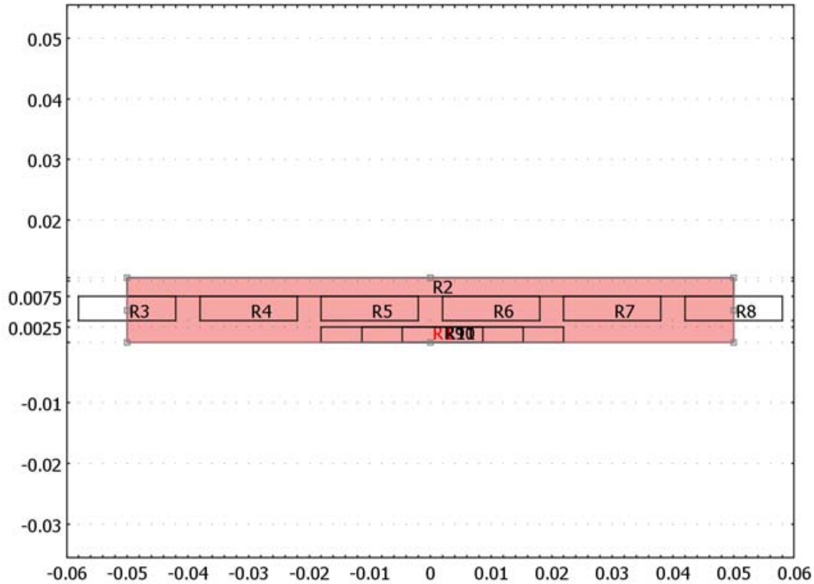
4 Select R3 and click the **Array** button. Set the **Displacement** in the *x* direction to 0.02 and the **Array size** in the *x* direction to 6.

5 Draw a rectangle R9 with opposite corners at (–0.018, 0) and (0.022, 0.0025).

6 Copy R9 into two new rectangles, pasting them at the same location.

7 Select R11 and click the **Scale** button on the draw toolbar. If it is difficult to select R11, a useful trick is to click the **Create Composite Object** button in the vertical toolbar, select the desired object in the **Object selection** list and click **Cancel**. Set the **Scale factor** in the *x* direction to `2/3` and the **Scale factor** in the *y* direction to `1` and change the **Scale base point** to (0.002, 0).

8 Select R10 and click the **Scale** button on the Draw toolbar. Set the **Scale factor** in the x direction to `1/3` and the **Scale factor** in the y direction to `1` and change the **Scale base point** to (0.002, 0).



9 Press Ctrl+A to select all objects and then open the **Create Composite Object** dialog box. Enter the expression `(R1+...R11)*R1` in the **Set formula** edit field.

10 Draw a line L1 from (0.002, 0) to (0.002, 0.0025).

PHYSICS SETTINGS

Boundary Conditions

Specify boundary conditions according to the following table.

SETTINGS	BOUNDARIES 2, 16, 22, 25, 30, 36, 39, 44	ALL OTHERS
Boundary condition	Electric insulation	Magnetic insulation

Subdomain Settings

Specify the subdomain properties according to the following table.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 3	SUBDOMAINS 2, 6, 13	SUBDOMAINS 4, 10, 14
$\mathbf{B} \leftrightarrow \mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$	$\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$
σ	`0`	`sigFe`	`sigFe`	`sigFe`

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 3	SUBDOMAINS 2, 6, 13	SUBDOMAINS 4, 10, 14
μ_r	1	muFe	1	1
$\mathbf{B}_r$	0 0	0 0	0 Br	0 -Br
$\mathbf{J}^e$	0	0	0	0

SETTINGS	SUBDOMAINS 5, 7	SUBDOMAINS 9, 11	SUBDOMAINS 8, 12
$\mathbf{B} \leftrightarrow \mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$
σ	sigCu	sigCu	sigCu
μ_r	1	1	1
$\mathbf{B}_r$	0 0	0 0	0 0
$\mathbf{J}^e$	Jin	-Jin	0

MESH GENERATION

Click the **Initialize Mesh** button on the Main toolbar.

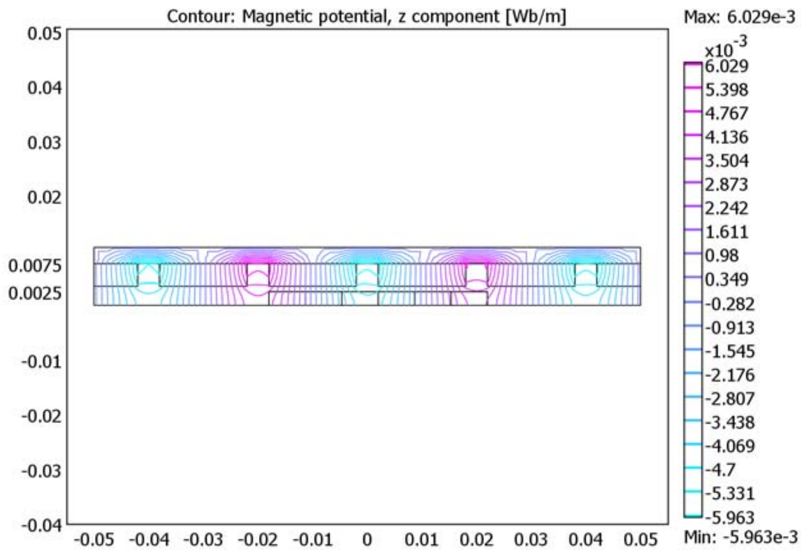
COMPUTING THE SOLUTION

1 Select the **Adaptive mesh refinement** check box in the **Solver Parameters** dialog box.

2 Click the **Solve** button on the Main toolbar to start the simulation.

POSTPROCESSING AND VISUALIZATION

1 In the **Plot Parameters** dialog box, select **Contour** plot and select the magnetic potential as **Contour data**. On the **Contour** page set the **Colormap** to **cool**.



From the results obtained with the simulation you can compute the force on the coils with `cemforce` from the command line. This requires that you run COMSOL Multiphysics together with COMSOL Script or MATLAB.

1 From the **File** menu select **Export>FEM Structure as 'fem'**.
2 Give the following command at the prompt:

```
F = cemforce(fem,'Wm_emqa','dl',[5 7 8 9 11 12])
```

This gives the result

```
F =
 -140.5877
 3.8e+003
```

Note that only the force in the x direction makes sense in this case. The reason is that when moving the coils in the y direction, the total area of the simulation structure changes, which causes the energy balance to break down.

3 In this case you can also obtain the forces using the Lorentz expression

$$\mathbf{F} = \mathbf{J} \times \mathbf{B} = \left[-J_{\text{in}} \cdot B_{y,}\ J_{\text{in}} \cdot B_x, 0\right].$$

Compute the total integrated force in the x direction with the command

```
Fx=postint(fem,'-Jez_emqa*By_emqa','dl',[5 7 8 9 11 12])
```

This gives the result

```
Fx =

 -137.7809
```

Because the model only includes half of the linear motor, you must multiply the results by a factor of 2 to obtain the total force density.

Modeling Using the Programming Language

```
% Clear the FEM and application structures.
clear fem appl

% Define the FEM structure constants.
fem.const = {...
  'sigCu',5.99e7,...
  'sigFe',1.03e7,...
  'muFe',2e3,...
  'Br0',1.23,...
  'Jin',3.66e6};

% Create the geometry by first setting all solid objects in a cell
% array and then calling geomcsg.
ss = cell(6,1);
ss{1} = rect2(-0.05,0.05,0,0.0105);
ss{2} = rect2(-0.05,0.05,0.0075,0.0105);
dx = 0;
for i1 = 3:8
  ss{i1} = rect2(-0.058+dx,-0.042+dx,0.0035,0.0075);
  dx = dx+0.02;
end
ss{9} = rect2(-0.018,0.022,0,0.0025);
ss{10} = scale(ss{9},2/3,1,0.002,0);
ss{11} = scale(ss{9},1/3,1,0.002,0);
cc{1} = curve2([0.002 0.002],[0 0.0025]);
g1 = solid2(geomcsg(ss,cc));
fem.geom = solid2(g1*ss{1});
fem.sdim = {'x' 'y'};

% Create the application structure.
appl.mode = 'PerpendicularCurrents';
appl.bnd.type = {'A0' 'tH0'};
```

```
appl.bnd.ind = ones(1,55);
appl.bnd.ind([2 16 22 25 30 36 39 44]) = 2;
appl.equ.sigma = {'0' 'sigFe' 'sigFe' 'sigFe'...
  'sigCu' 'sigCu' 'sigCu'};
appl.equ.mur = {'1' 'muFe' '1' '1'...
  '1' '1' '1'};
appl.equ.Br = {{'0' '0'} {'0' '0'} {'0' 'Br0'}...
 {'0' '-Br0'} {'0' '0'} {'0' '0'} {'0' '0'}};
appl.equ.magconstrel = {'mur' 'mur' 'Br' 'Br' 'mur' 'mur' 'mur'};
appl.equ.Jez={'0' '0' '0' '0'...
  'Jin' '-Jin' '0'};
appl.equ.ind = [1 3 2 4 5 3 5 7 6 4 6 7 3 4];
appl.prop.elemdefault = 'lag1';

% Generate the PDE coefficients and boundary conditions for the
% FEM structure.
fem.appl = appl;
fem = multiphysics(fem);

% Generate the mesh.
fem.mesh = meshinit(fem);
fem.xmesh = meshextend(fem);

% Solve the problem using the adaptive solver with two
% refinements.
fem = adaption(fem,'ngen',3,'report','on');

% Visualize the solution.
postplot(fem,'contdata','Az','contmap','cool','geom','on')

% You can compute the forces in the same way as for the GUI-based
% analysis.
F = cemforce(fem,'Wm','dl',[5 7 8 9 11 12])
Fx = postint(fem,'-Jez*By','dl',[5 7 8 9 11 12])
```

Generator in 2D

Introduction

This example shows how the circular motion of a rotor with permanent magnets generates an induced EMF in a stator winding. The generated voltage is calculated as a function of time during the rotation. The model also shows the influence on the voltage from material parameters, rotation velocity, and number of turns in the winding.

The center of the rotor consists of annealed medium carbon steel, which is a material with a high relative permeability. The center is surrounded with several blocks of a permanent magnet made of samarium and cobalt, creating a strong magnetic field. The stator is made of the same permeable material as the center of the rotor, confining the field in closed loops through the winding. The winding is wound around the stator poles. Figure 3-1 shows the generator with part of the stator sliced in order to show the winding and the rotor.

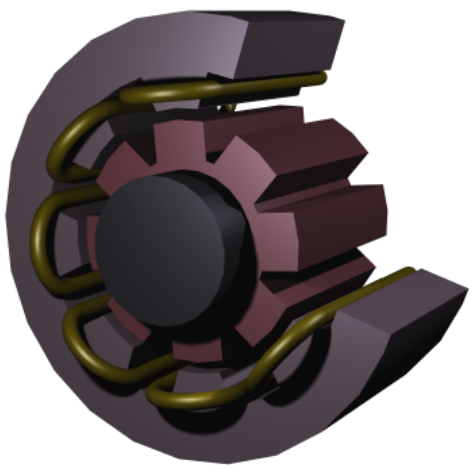


Figure 3-1: Drawing of a generator showing how the rotor, stator, and stator winding are constructed. The winding is also connected between the loops, interacting to give the highest possible voltage.

Modeling in COMSOL Multiphysics

The COMSOL Multiphysics model of the generator is a time-dependent 2D problem on a cross section through the generator. This is a true time-dependent model where the motion of the magnetic sources in the rotor is accounted for in the boundary condition between the stator and rotor geometries. Thus, there is no Lorentz term in the equation, resulting in the PDE

$$\sigma\frac{\partial \mathbf{A}}{\partial t} + \nabla \times \left(\frac{1}{\mu}\nabla \times \mathbf{A}\right) = 0$$

where the magnetic vector potential only has a z component.

Rotation is modeled using a deformed mesh application mode (ALE), in which the center part of the geometry, containing the rotor and part of the air-gap, rotates with a rotation transformation relative to the coordinate system of the stator. The rotation of the deformed mesh is defined by the transformation

$$\begin{bmatrix} x_{\mathrm{rot}} \\ y_{\mathrm{rot}} \end{bmatrix} = \begin{bmatrix} \cos(\omega t) & -\sin(\omega t) \\ \sin(\omega t) & \cos(\omega t) \end{bmatrix} \cdot \begin{bmatrix} x_{\mathrm{stat}} \\ y_{\mathrm{stat}} \end{bmatrix}$$

The rotor and the stator are drawn as two separate geometry objects, so it is possible to use an assembly (see "Using Assemblies" on page 351 of the *COMSOL Multiphysics Modeling Guide* for further details). This has several advantages: the coupling between the rotor and the stator is done automatically, the parts are meshed independently, and it allows for a discontinuity in the vector potential at the interface between the two geometry objects (called slits). The rotor problem is solved in a rotating coordinate system where the rotor is fixed (the rotor frame), whereas the stator problem is solved in a coordinate system that is fixed with respect to the stator (the stator frame). An identity pair connecting the rotating rotor frame with the fixed stator frame is created between the rotor and the stator. The identity pair enforces continuity for the vector potential in the global fixed coordinate system (the stator frame).

The material in the stator and the center part of the rotor has a nonlinear relation between the magnetic flux, $\mathbf{B}$ and the magnetic field, $\mathbf{H}$, the so called B-H curve. In COMSOL Multiphysics, the B-H curve is introduced as an interpolation function; see Figure 3-2. The function can be used in the subdomain settings. Usually B-H curves are specified as $|\mathbf{B}|$ versus $|\mathbf{H}|$, but the perpendicular waves application mode must have $|\mathbf{H}|$ versus $|\mathbf{B}|$. It is therefore important that the H-data is entered as $f(x)$-data of the interpolation function and the B-data entered as x-data. This relationship for $|\mathbf{H}|$ is predefined for the material **Soft Iron (without losses)** in the materials library that is shipped with the AC/DC Module, `acdc_lib.txt`.

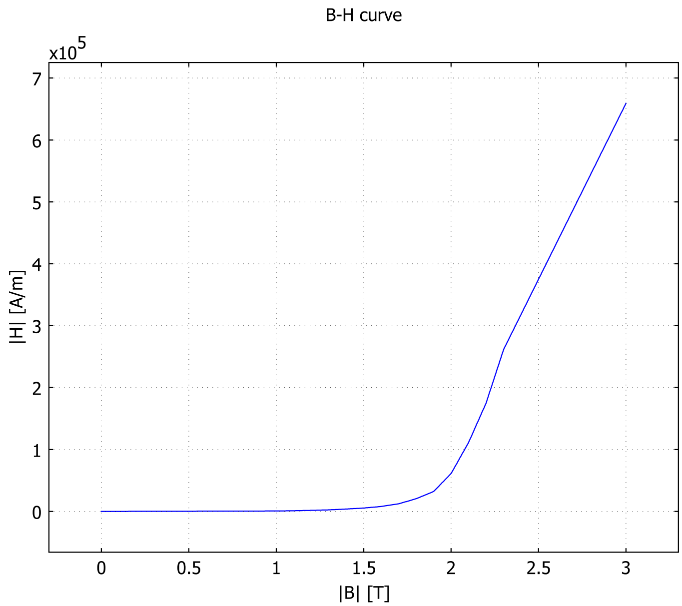


Figure 3-2: The norm of the magnetic flux, $|\mathbf{B}|$*, versus the norm of the magnetic field,* $|\mathbf{H}|$*, for the rotor and stator materials. The inverse of this curve is used in the calculation.*

The generated voltage is computed as the line integral of the electric field, **E**, along the winding. Because the winding sections are not connected in the 2D geometry, a proper line integral cannot be carried out. A simple approximation is to neglect the voltage contributions from the ends of the rotor, where the winding sections connect. The voltage is then obtained by taking the average z component of the **E** field for each winding cross-section, multiplying it by the axial length of the rotor, and taking the sum over all winding cross sections.

$$V_i = \mathrm{NN} \sum_{\text{windings}} \frac{L}{A} \int E_z dA$$

where L is the length of the generator in the third dimension, NN is the number of turns in the winding, and A is the total area of the winding cross-section.

Results and Discussion

The generated voltage in the rotor winding is a sinusoidal signal. At a rotation speed of 60 rpm the voltage will have an amplitude around 2.3 V for a single turn winding; see Figure 3-3.

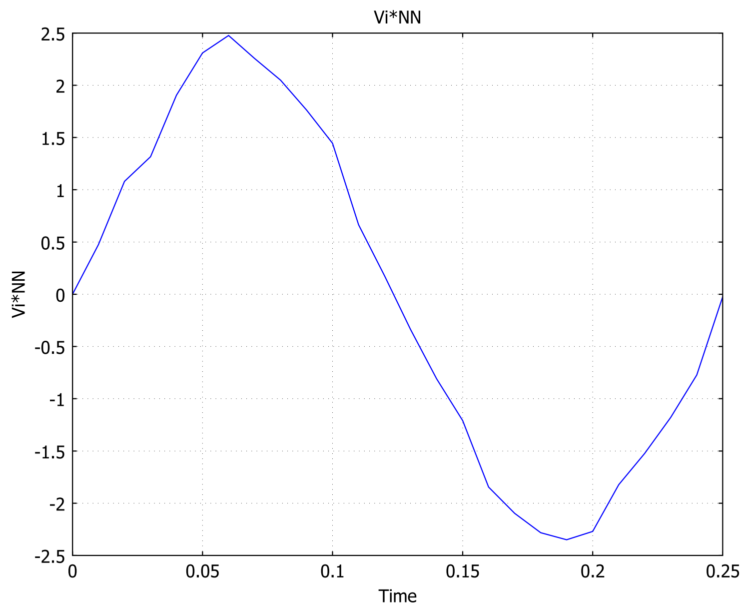


Figure 3-3: The generated voltage over one quarter of a revolution. This simulation used a single-turn winding.

The norm of the magnetic flux, |**B**|, and the field lines of the **B** field are shown below in Figure 3-4 at time 0.20 s.

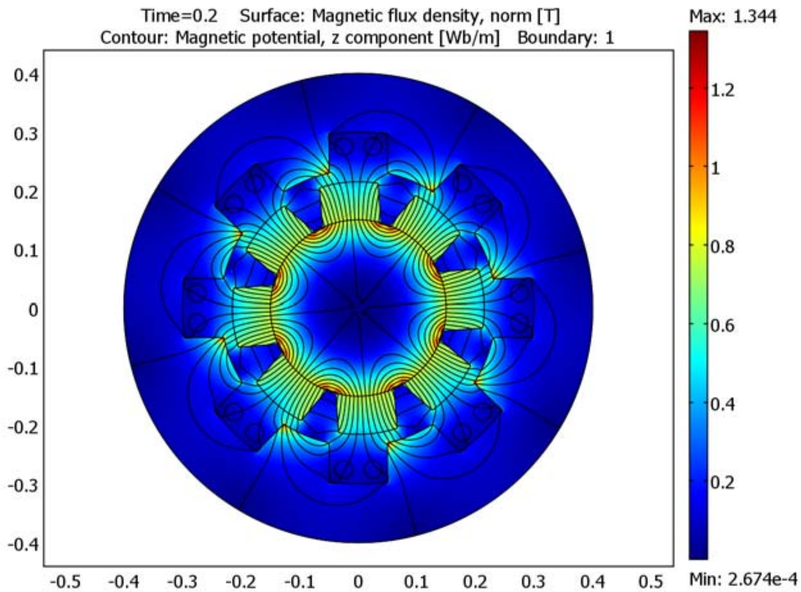


Figure 3-4: The norm and the field lines of the magnetic flux after 0.2 s of rotation. Note the brighter regions, which indicate the position of the permanent magnets in the rotor.

Model Library path: `ACDC_Module/Motors_and_Drives/generator_2d`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator**, make sure that **2D** is selected in the **Space dimension** list.

2 In the **AC/DC Module** folder select **Rotating Machinery>Rotating Perpendicular Currents**.

3 Click **OK** to close the **Model Navigator**.

OPTIONS AND SETTINGS

1 From the **Options** menu choose **Constants**.

2 In the **Constants** dialog box define the following constants with names and expressions.The description field is optional. When done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`t`	`0[s]`	`Time for stationary solution`
`rpm`	`60[1/min]`	`Revolutions per minute`
`A`	`pi*(0.02[m])^2`	`Area of wires in the stator`
`L`	`0.4[m]`	`Length of the generator`
`NN`	`1`	`Number of turns in stator winding`

When you enter the constant `t` in the table, the text turns red. The reason is that `t` is the reserved variable name for time in COMSOL Multiphysics. However, because of the solution procedure in this model—you first compute a stationary solution which you then use as the initial condition for a subsequent time-dependent simulation—`t` needs to be given a value (zero) also for the stationary case. It will subsequently be overridden by the `t` values from the transient solver.

GEOMETRY MODELING

The geometry modeling of this model is rather extensive, so you can choose to import the generator geometry from a binary file. Select the section that you prefer.

Importing the Geometry from a Binary File

1 From the **File** menu, select **Import>CAD Data From File**.

2 In the **Import CAD Data From File** dialog box, make sure that either **All 2D CAD files** or **COMSOL Multiphysics file** is selected in the **Files of type** list.

3 From the COMSOL Multiphysics installation directory, go to the model library path given on page 43. Select the `generator_2d.mphbin` file, and click **Import**.

4 Skip the section "Creating the Geometry from Scratch" and continue at "Assemble the Geometry" on page 47.

Creating the Geometry from Scratch

Please follow the steps in the geometry modeling carefully. In the following sections, it is assumed that the boundary numbering is that resulting from these steps. After all geometry operations and the assembly step, there should be a total of 152 boundaries.

1 Choose **Draw>Specify Objects>Circle** to create circles with the following properties:

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
C1	`0.3`	Center	0	0	`22.5`
C2	`0.235`	Center	0	0	0

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
C3	0.225	Center	0	0	0
C4	0.4	Center	0	0	0

2 Choose **Draw>Specify Objects>Rectangle** to create rectangles with the following properties:

NAME	WIDTH	HEIGHT	BASE	X	Y	ROTATION ANGLE
R1	0.1	1	Center	0	0	0
R2	0.1	1	Center	0	0	45
R3	0.1	1	Center	0	0	90
R4	0.1	1	Center	0	0	135

3 Open the **Create Composite Object** dialog box from the **Draw** menu or click on the toolbar button with the same name.

4 In the dialog box clear the **Keep interior boundaries** check box.

5 Type `C2+C1*(R1+R2+R3+R4)` in the **Set formula** edit field.

6 Click **OK**.

7 Open the **Create Composite Object** dialog box again and select the **Keep interior boundaries** check box.

8 Type `C4+CO1-C3` in the **Set formula** edit field.

9 Click **OK**.

10 Create two new circles according to the table below.

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
C1	0.015	Center	0.025	0.275	0
C2	0.015	Center	-0.025	0.275	0

11 Click the **Zoom Extents** button to get a better view of the geometry.

12 Select the circles C1 and C2.

13 Press Ctrl+C to copy the circles.

14 Press Ctrl+V to paste the circles. Make sure that the displacements are zero and click **OK**.

15 Go to **Draw>Modify>Rotate** and type 45 in the **Rotation angle** edit field before clicking **OK**.

16 Repeat the last two steps but with rotation angles 90, 135, 180, –45, –90, and –135.

17 Select all objects by pressing Ctrl+A, then click the **Union** toolbar button.

18 Create four circles with the following properties:

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
C1	0.215	Center	0	0	0
C2	0.15	Center	0	0	22.5
C3	0.15	Center	0	0	22.5
C4	0.225	Center	0	0	0

19 Create four rectangles with the following properties:

NAME	WIDTH	HEIGHT	BASE	X	Y	ROTATION ANGLE
R1	0.1	1	Center	0	0	22.5
R2	0.1	1	Center	0	0	-22.5
R3	0.1	1	Center	0	0	67.5
R4	0.1	1	Center	0	0	-67.5

20 Open the **Create Composite Object** dialog box and clear the **Keep interior boundaries** check box.

21 Type `C2+C1*(R1+R2+R3+R4)` in the **Set formula** edit field.

22 Click **OK**.

23 Open the **Create Composite Object** dialog box again and select the **Keep interior boundaries** check box.

24 Type `CO2+C3+C4` in the **Set formula** edit field.

25 Click **OK**.

The geometry in the drawing area should now look like that in Figure 3-5.

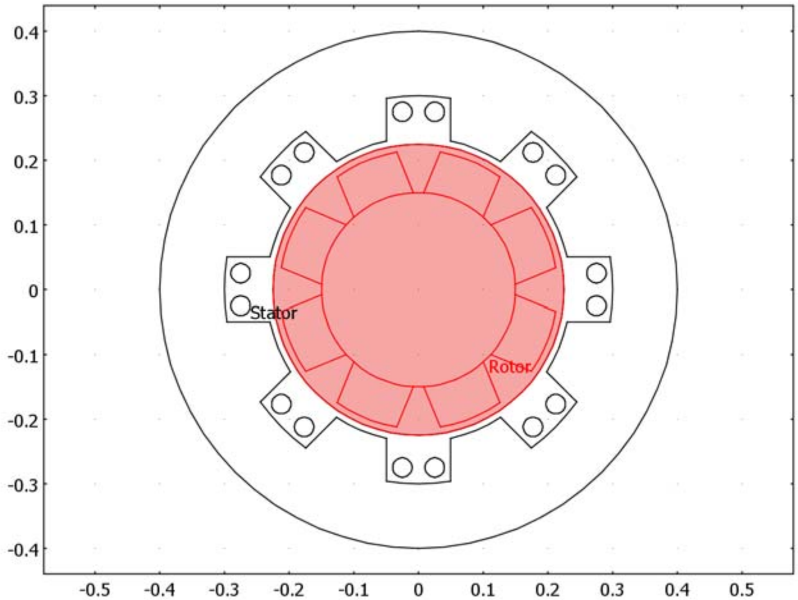


Figure 3-5: Generator geometry.

Assemble the Geometry

1 Open the **Create Pairs** dialog box from the **Draw** menu.

2 Select both objects (CO1 and CO3), and click **OK**.

You have now created two separate geometry objects in an assembly. You have also created an identity pair connecting the physics in the rotor and the stator, taking the rotation into account. An interface between two assemblies is actually two groups of overlapping boundaries that are, without the pair connection, completely separated in terms of mesh and solution variables. This is necessary to allow a sliding mesh at the interface between the stator and rotor where the solution variable becomes discontinuous across that interface.

PHYSICS SETTINGS

1 From the **Options** menu point to **Integration Coupling Variables** and click **Subdomain Variables**.

2 In the **Subdomain Integration Variables** dialog box define a variable with integration order 4 and global destination according to the table below.

NAME	EXPRESSION IN SUBDOMAINS 5–8, 13–16	EXPRESSION IN SUBDOMAINS 3, 4, 9–12, 17, 18
`Vi`	`-L*Ez_emqa/A`	`L*Ez_emqa/A`

3 Click **OK**.

Steps 4–6 below are only necessary to check that the library you need is loaded and to view the interpolation function of one of the materials in this library. These steps can be skipped if you know that you have the library loaded. If you later notice that you are missing this library, just go back to these steps and add the library.

4 Open the **Options** menu and select **Materials/Coefficients Library**.

5 Make sure that the library `acdc_lib.txt` is loaded (click the minus in front of other libraries to remove their content from the view). If you cannot find the library, load the library by clicking the **Add Library** button. The file should be located under the `data` folder in the COMSOL installation directory.

6 Click **OK** to close the **Materials/Coefficients Library** dialog box.

Subdomain Settings

1 From the **Multiphysics** menu, select **Perpendicular Induction Currents, Vector Potential (emqa)** if it is not already selected.

2 Choose **Physics>Properties**.

3 In the **Application Mode Properties** dialog box, choose **On** from the **Weak constraints** list. The ideal weak constraints help conserving the magnetic flux across the assembly boundaries between the rotor and the stator. This allows you to specify an absolute tolerance for this entity when solving the time-dependent model.

4 Click **OK**.

5 From the **Physics** menu open the **Subdomain Settings**, then select any entry in the **Subdomain selection** list.

6 Click the **Load** button to open the **Materials/Coefficients Library** dialog box.

7 In the dialog box select the **Soft Iron (without losses)** material from the **Electric (AC/DC Material Properties** library.

8 Click **Apply**. The material **Soft Iron (without losses)** has now been added to the model. You see which materials you have added to the model under the **Model** item in the **Materials** tree view.

9 Repeat the last two steps for the materials **Samarium Cobalt (Radial, inward)** and **Samarium Cobalt (Radial, outward)**. Before you close the dialog box, make sure that you see all three materials under **Model** in the **Materials** tree view.

10 Click **Cancel** to close the **Materials/Coefficients Library** dialog box.

11 Enter subdomain settings and select materials according to the table below. To change the constitutive relation, you need to choose the relation from the **Constitutive relation** list.

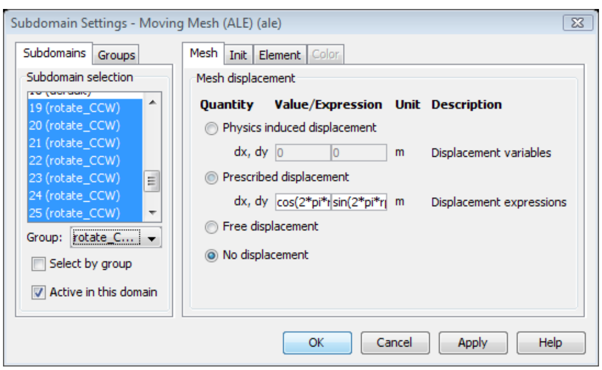
SETTINGS	SUBDOMAINS 20, 23, 24, 27	SUBDOMAINS 21, 22, 25, 26	SUBDOMAINS 2, 28	ALL OTHER SUBDOMAINS
$\mathbf{B} \leftrightarrow \mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$	$\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$	$\mathbf{H} = f(\lvert\mathbf{B}\rvert)\mathbf{e}_B$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$
Library material	Samarium Cobalt (Radial, inward)	Samarium Cobalt (Radial, outward)	Soft Iron (without losses)	

12 Click **OK**.

13 From the **Multiphysics** menu select **Moving Mesh (ALE) (ale)**.

14 From the **Physics** menu open the **Subdomain Settings**.

15 Select Subdomains 19–28, then select the predefined group **rotate_CCW** from the **Group** list.



This group uses a prescribed displacement with expressions that define counterclockwise rotation around the origin. The default setting for all other subdomains is no displacement.

16 Click **OK** to close the dialog box.

Boundary Conditions

Leave the boundary settings at their defaults. The identity pair creates a coupling between the rotor and the stator in the ALE frame.

MESH GENERATION

1 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.
2 From the **Predefined mesh sizes** list select **Finer**.
3 Click the **Custom mesh size** option button and type 2 in the **Resolution of narrow regions** edit field.
4 Click **Remesh** to generate the mesh.
5 Click **OK** to close the dialog box.

COMPUTING THE SOLUTION

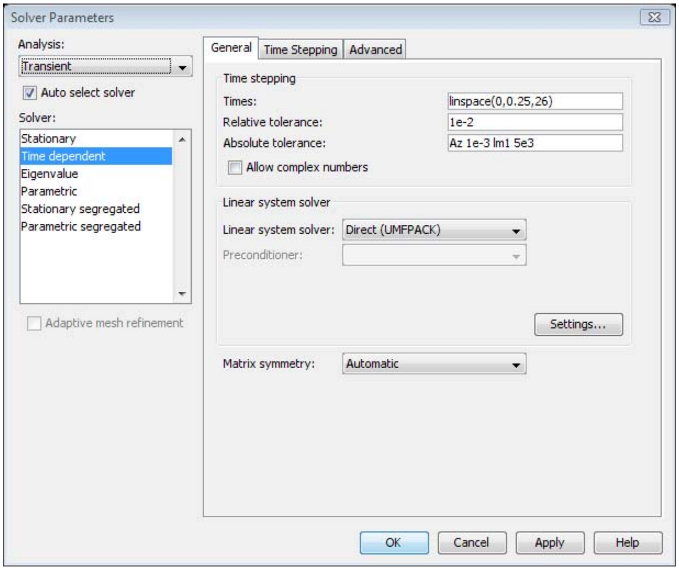
Begin by computing the initial solution for the subsequent time-dependent simulation.

1 Open the **Solver Parameters** dialog box by clicking the corresponding button on the Main toolbar or from the **Solve** menu.
2 With the **Auto select solver** check box selected, select **Static** from the **Analysis** list.
3 Click **OK**.
4 Click the **Solve** button.

Next, simulate the generator with the rotor in motion. By default, the solution you just calculated will be used as the initial solution.

1 Open the **Solver Parameters** dialog box again and select **Transient** from the **Analysis** list.

2 In the **Times** edit field type `linspace(0,0.25,26)`, and in the **Absolute tolerance** edit field type `Az 1e-3 lm1 5e3`.



3 Click **OK**.

4 Click the **Restart** button.

POSTPROCESSING AND VISUALIZATION

1 From the **Postprocessing** menu open the **Plot Parameters** dialog box.

2 On the **General** page select the solution for time 0.2 s from the **Solution at time** list. Clear the **Geometry edges** check box.

3 On the **Surface** page select **Magnetic flux density, norm** from the **Predefined quantities** list or type `normB_emqa` in the **Expression** edit field.

4 On the **Contour** page type `Az` in the **Expression** edit field and type `15` in the **Levels** edit field. Click to select the **Contour plot** check box.

5 In the **Contour color** area click the **Uniform color** option button, and clear the **Color scale** check box.

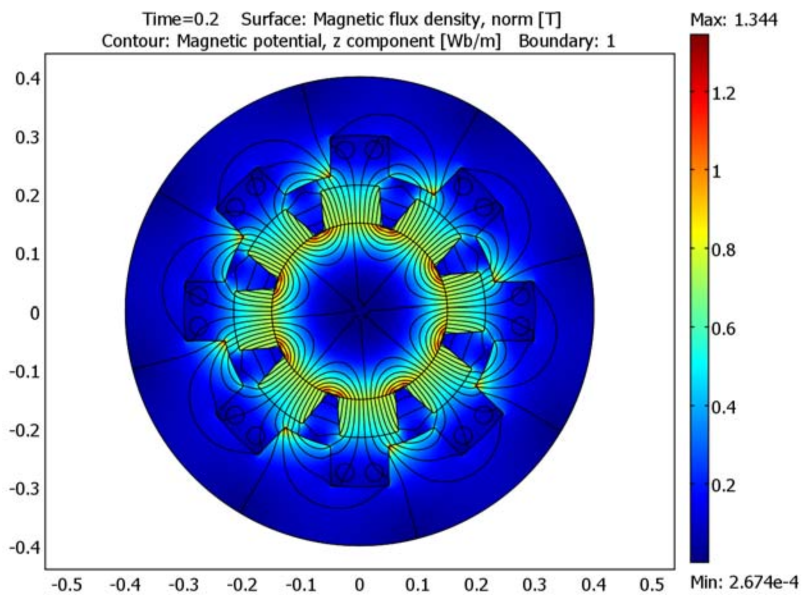
6 Click the **Color** button and select a black color. Click **OK** to close the **Contour Color** dialog box.

7 On the **Boundary** page type `1` in the **Expression** edit field. Click to select the **Boundary plot** check box.

8 In the **Boundary color** area click the **Uniform color** option button.

9 Click the **Color** button and select a black color. Click **OK** to close the **Boundary Color** dialog box.

10 Click **OK** to close the **Plot Parameters** dialog box.



To view the induced voltage as a function of time, proceed with the steps below.

1 From the **Postprocessing** menu, select **Domain Plot Parameters**.

2 On the **Point** page select an arbitrary point in the **Point selection** list, then type `Vi*NN` in the **Expression** edit field.

3 Click **OK**.

To create an animation of the rotating rotor, do the following steps.

1 Open the **Plot Parameters** dialog box.

2 Click the **Animate** tab and make sure that all time steps are selected.

3 Click the **Start Animation** button. When the movie has been generated, the *COMSOL Movie Player* opens it.

Generator with Mechanical Dynamics and Symmetry

Introduction

This is an extension of the generator model including an ordinary differential equation for the mechanical dynamics and computation of the torque resulting from the magnetic forces. In addition it exploits symmetry to reduce the model size to 1/8 of the original size. The model also involves a lumped winding resistance.

Modeling in COMSOL Multiphysics

The reader is supposed to be familiar with the generator model in the preceding section. Here the focus is on concepts that the previous model does not cover. The smallest possible model of the generator is a sector obtained by cutting radially through two adjacent rotor poles, see Figure 3-6.

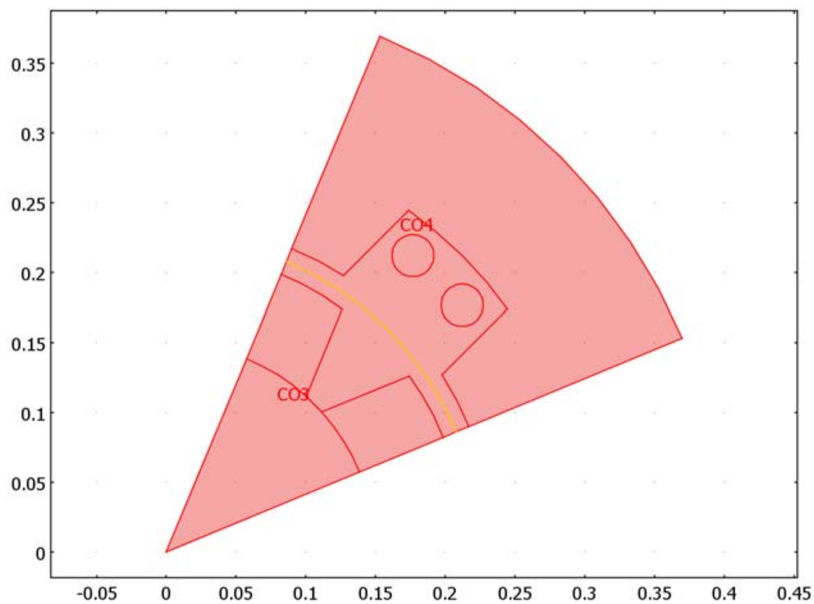


Figure 3-6: The smallest possible sector model.

Because adjacent rotor poles are magnetized radially but in opposite directions, such a sector model exhibits antisymmetry. This is readily accounted for in COMSOL Multiphysics by the support for periodic boundary conditions. In addition, the sliding mesh coupling must be modified to keep the coupling coordinates within the modeled geometry sector. There is a special sliding mesh condition for such symmetry or antisymmetry connections that defines one coupling operator for each sector in the model. In this case there are 8 such coupling operators. The first operator just performs a direct coupling between the rotor and stator. After a little less than half revolution for example, the third and fourth operator perform the coupling. This special sector symmetry condition requires that you use assemblies with the Moving Mesh application mode, similar to how it is done in the preceding generator model. The main difference with that model, is that you will use the fact that the magnetic fields do not change between the reference and moving frame, and that the mesh in the moving domains are not deformed, only rotated. As a consequence, you can turn off the moving mesh transformations for the electromagnetic application mode and use them only for the coupling operators. You save a significant amount of execution time by not performing the mesh transformations, and the result is the same.

The ODE for the mechanical dynamics

$$\frac{d^2\alpha}{dt^2} = \frac{T_z + M}{\mathrm{I}_0 + \mathrm{I}_{\mathrm{rotor}}}$$

involves the torque $T_z + M$ and the moment of inertia $\mathrm{I}_0 + \mathrm{I}_{\mathrm{rotor}}$. M is the externally applied driving torque and T_z is the electromagnetic torque. The model computes the latter automatically by boundary integration of the cross product between the radius vector (x,y) and the normal Maxwell stress components (nT_x, nT_y) over the rotor boundary. Because the model depth is not unity, the torque variable must be multiplied with the variable L, and a factor 8 to account for the entire device.

$$T_z = 8\mathrm{L}T_z^{(\mathrm{auto})}$$

The moment of inertia is composed of an external moment of inertia, I_0 (flywheel and turbine), and the moment of inertia of the rotor itself, $\mathrm{I}_{\mathrm{rotor}}$. The model computes the latter by subdomain integration of the material density ρ times radius vector squared times the model depth L. Again, the factor 8 accounts for the entire generator.

$$\mathrm{I}_{\mathrm{rotor}} = \iint_A 8\rho\mathrm{L}(x^2 + y^2)dA$$

The model accounts for the lumped winding resistance by setting the winding current to the induced voltage divided by the winding resistance. This corresponds to a generator operating with the output terminals in short circuit. The model computes the induced voltage using a subdomain integration coupling variable.

Results and Discussion

The startup of the generator was simulated for 2 s which is sufficient for reaching steady state. The time evolution of the rotation angle is shown in Figure 3-7.

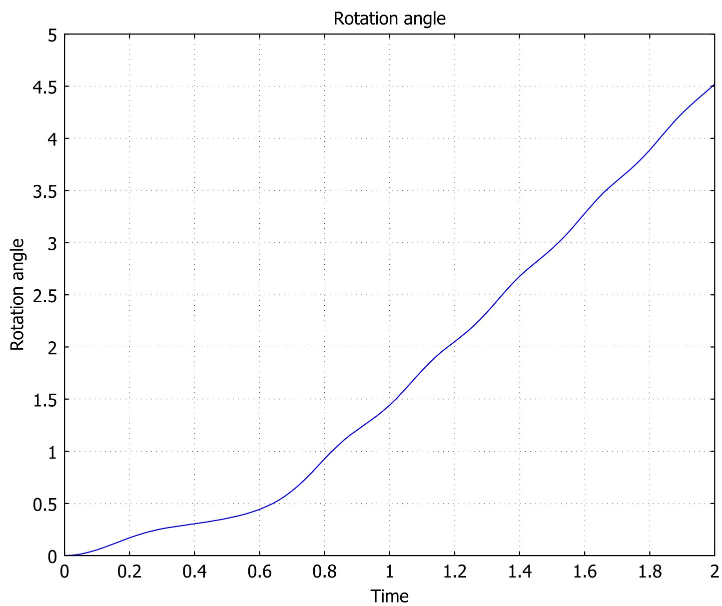


Figure 3-7: The time evolution of the rotation angle is shown. After about 1 s, the rotation angle increases linearly with time corresponding to constant rotation speed.

Model Library path: `ACDC_Module/Motors_and_Drives/generator_sector_dynamic`

MODEL NAVIGATOR

1 In the **Model Navigator**, make sure that **2D** is selected in the **Space dimension** list.

2 In the **AC/DC Module** folder select **Rotating Machinery>Rotating Perpendicular Currents**.

3 In the **Dependent variables** edit field, enter `Az X Y Z`.

4 Click **OK** to close the **Model Navigator**.

OPTIONS AND SETTINGS

1 From the **Options** menu choose **Constants**.

2 In the **Constants** dialog box define the following constants with names and expressions. When done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
A	pi*(0.02[m])^2	Area of wires in the stator
L	0.4[m]	Length of the generator
NN	1	Number of turns in stator winding
M	1400[N*m]	Externally applied torque
Rc	1e-4[ohm]	Resistance of winding
I0	100[kg*m^2]	External moment of inertia

GEOMETRY MODELING

1 From the **File** menu, select **Import>CAD Data From File**.

2 In the **Import CAD Data From File** dialog box, make sure that **All 2D CAD files** or **COMSOL Multiphysics file** is selected in the **Files of type** drop-down menu.

3 From the COMSOL Multiphysics installation directory, browse to the folder given on page 55. Select the file `generator_2d.mphbin`, then click **Import**.

4 Choose **Draw>Specify Objects>Line** and select **Closed polyline (solid)** in the **Style** list. Enter **Coordinates x**: `0 0.4*cos(67.5*pi/180) 0.4 0.4*cos(22.5*pi/180) 0` and **Coordinates y:** `0 0.4*sin(67.5*pi/180) 0.4 0.4*sin(22.5*pi/180) 0`.

5 Click **OK** to create the composite object CO3.

6 Press Ctrl+C to copy the created object.

7 Select the objects (CO1 and CO3), then click the **Intersection** button on the Draw toolbar to create the geometry object, CO4.

8 Press Ctrl+V to paste the polyline object, just click **OK** in the dialog box that appears. Then select the pasted object and the second imported object (CO1 and CO2).

9 Click the **Intersection** button on the Draw toolbar.

10 Click the **Zoom Extents** button on the Main toolbar. You now have a 1/8 sector of the original generator model geometry, with one object for the rotor and one for the stator.

11 Select both remaining objects and click the **Create Pairs** button on the Draw toolbar.

12 From the **Physics** menu, choose **Identity Pairs>Identity Boundary Pairs**.

13 Select Pair 1 and click the **Interchange source and destination** button between the **Source boundaries** and **Destination boundaries** fields.

14 Click **OK**. It is important that the source side of the assembly pair is on the fixed side, and that the destination side is on the moving side. Otherwise the solver does not detect the motion.

PHYSICS SETTINGS

Global Expressions

1 From the **Options** menu, point to **Expressions** and click **Global Expressions**.

2 In the **Global Expressions** dialog box, define the following expression:

NAME	EXPRESSION	DESCRIPTION
`Tz`	`8*L[1/m]*F_torquez_emqa`	`Total torque for the entire device`

3 Click **OK**. The unit specification in the expression is there to keep the correct unit of the torque.

Subdomain Expressions

1 From the **Options** menu, point to **Expressions** and click **Subdomain Expressions**.

2 In the **Subdomain Expressions** dialog box, define the following expressions:

NAME	EXPRESSION IN SUBDOMAINS 1–4	EXPRESSION IN ALL OTHER SUBDOMAINS
`u`	`-y*alphat`	`0`
`v`	`x*alphat`	`0`

3 Add another variable `rho` for the material density and set it to `7870[kg/m^3]` in Subdomain 1 and to `8400[kg/m^3]` in Subdomains 2 and 4.

4 Click **OK**.

Integration Coupling Variables

1. In the **Options** menu, point to **Integration Coupling Variables** and select **Subdomain Variables**.
2. In the **Subdomain Integration Variables** dialog box, define variables according to the table below with default global destination and default integration order 4.

NAME	EXPRESSION IN SUBDOMAINS 1, 2, 4	EXPRESSION IN SUBDOMAINS 7, 8
`Vi`		`8*L*NN*Ez_emqa/A`
`I`	`8*L*rho*(x^2+y^2)`	

3. Click **OK**.

Because of the use of coupling variables, COMSOL Multiphysics cannot determine the unit for other variables that directly or indirectly depend on these coupling variables. This causes the warnings for inconsistent units in the specification of the magnetic potential and external current density. You can disregard these warnings.

Subdomain Settings (Moving Mesh Application Mode)

1. From the **Physics** menu, open the **Subdomain Settings** dialog box.
2. Select Subdomains 1–4. From the **Group** list, select the predefined group called **angle_CCW**. This group uses the variable named omega as a counter-clock-wise rotation. You later declare this variable as an ODE variable.
3. Click **OK** to close the dialog box.

Application Mode Properties (Perpendicular Induction Currents)

1. From the **Multiphysics** menu, select **Perpendicular Induction Currents, Vector Potential (emqa)** to switch application mode.
2. From the **Physics** menu, choose **Properties**.
3. Choose **On** from the **Weak constraints** list and **Frame (ref)** from the **Frame** list.
4. Click **OK**.

Subdomain Settings

1. Open the **Subdomain Settings** dialog box from the **Physics** menu.
2. Select any entry from the **Subdomain selection** list, then click the **Load** button to open the **Materials/Coefficients Library** dialog box.
3. In the **Materials** tree, open the **Electric (AC/DC) Material Properties** library and select **Soft Iron (without losses)**.

4 Click **Apply**. The material **Soft Iron (without losses)** is added to the model. You see which materials you have added to the model under the **Model** item in the **Materials** tree view.

5 Repeat the two last steps for the materials **Samarium Cobalt (Radial, inward)** and **Samarium Cobalt (Radial, outward)**. Before you close the dialog box, make sure that you see all three materials under **Model** in the **Materials** tree view.

6 Click **Cancel** to close the **Materials/Coefficients Library** dialog box.

7 Enter subdomain settings and select materials according to the table below. To change the constitutive relation, you need to choose the relation from the **Constitutive relation** list.

SETTINGS	SUBDOMAIN 4	SUBDOMAIN 2	SUBDOMAIN 1, 6	ALL OTHERS
$\mathbf{B} \leftrightarrow \mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$	$\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$	$\mathbf{H} = f(\lvert\mathbf{B}\rvert)\mathbf{e}_B$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$
Library material	Samarium Cobalt (Radial, inward)	Samarium Cobalt (Radial, outward)	Soft Iron (without losses)	

8 Select Subdomains 7 and 8, and enter `NN*Vi/(Rc*A)` in the **External current density** edit field.

9 Click the **Forces** tab. Select Subdomains 1, 2, and 4 from the **Subdomain selection** list, then set the **Name** field to `F`. This setting generates variables for the force and torque on the selected subdomains. The torque variable gets the name `F_torquez_emqa`.

10 Click **OK**.

Boundary Conditions

1 In the **Boundary Settings** dialog box, enter the following settings:

SETTINGS	BOUNDARIES 1–4, 7, 8, 15, 16, 19, 20
Boundary condition	Periodic condition
Type of periodicity	Antiperiodicity

2 Click the **Weak Constr.** tab, select the Boundaries 1–4, 7, 8, 15, 16, 19–21 and 23, and clear the **Use weak constraints** check box. It is not necessary to use weak constraint on these boundaries, and they actually disturb the weak constraints on the sliding mesh boundaries if activated.

3 Click the **Pairs** tab and select Pair 1.

4 From the **Boundary condition** list, select **Sector antisymmetry**, then enter 8 in the **Number of sectors** edit field.

5 Click **OK**.

Periodic Point Conditions

You need to make the Lagrange multiplier variable lm1, associated with the weak boundary constraint, antisymmetric.

1 Choose **Physics>Periodic Conditions>Periodic Point Conditions**.

2 On the **Source** page, select Point 4 from the **Point selection** list, then enter `lm1` in the first row in the **Expression** column and press Return.

3 Click the **Source Vertices** tab, and click the **>>** button to use Point 4 as source vertex.

4 Click the **Destination** tab, then select Point 11.

5 Select the **Use selected points as destination** check box, then enter the expression `-lm1`.

6 Click the **Destination Vertices** tab, then click the **>>** button to add Point 11.

7 Click **OK**.

Space-Independent Equations

1 From the **Physics** menu, open the **Space-Independent Equations** dialog box and enter entities according to the following table. The description field is optional.

NAME	EQUATION	INIT(U)	INIT(UT)	DESCRIPTION
`omega`	`omegatt-(M+Tz)/(I+IO)`	`0`	`0`	`Rotation angle`

2 Click **OK**.

MESH GENERATION

You get the best performance for periodic boundary conditions if the mesh is identical for the source and destination boundaries of the periodic condition. The steps below use the Copy mesh feature to ensure an identical mesh.

1 In the **Free Mesh Parameters** dialog box, select the **Predefined mesh sizes** option button, then select **Finer** from the list.

2 On the **Boundary** page, select Boundaries 2, 5–8, 10–12, 14, and 19–21 from the **Boundary selection** list and enter a **Maximum element size** of `0.005`.

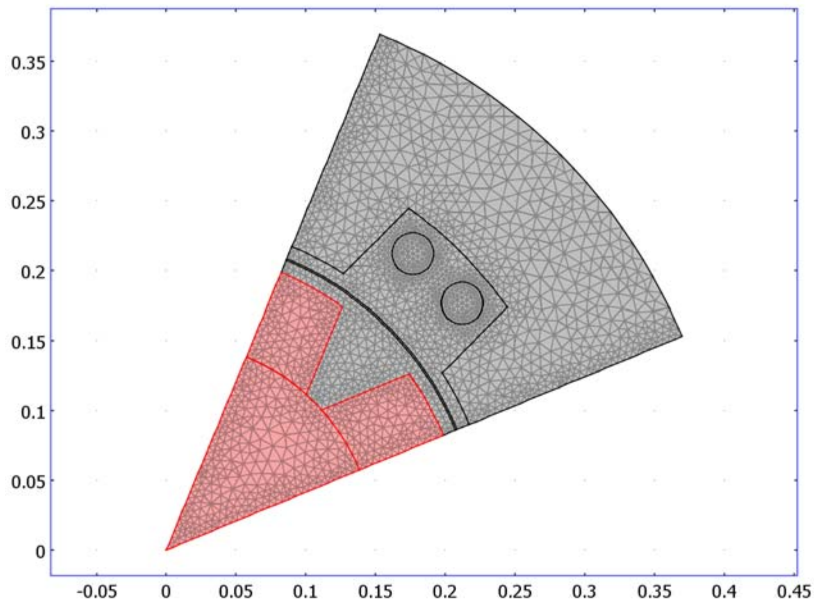
3 Click the **Mesh Selected** button.

4 Select Boundaries 1 and 2. The click the **Copy Mesh** button on the Mesh toolbar, which is located on the toolbar on the left side of the COMSOL Multiphysics window.

5 Repeat this step for the remaining four boundary pairs: 3 and 7, 4 and 8, 15 and 19, and 16 and 20. The mesh on the right side and the mesh on the left side are now identical.

6 Click **OK** to close the **Free Mesh Parameters** dialog box. This was only open under the copy mesh operation in order to have a numbered list of boundaries available.

7 Finally, click the **Mesh Remaining (Free)** button on the Mesh toolbar, to mesh the rest of the geometry. The resulting mesh looks like the one in the figure below.



COMPUTING THE SOLUTION

1 In the **Solver Parameters** dialog box, select **Stationary** from the **Solver** area.

2 Click **OK**.

3 Open the **Solver Manager** and go to the **Solve For** page.

4 Clear the variable **omega** under **ODE**.

5 Click **OK**.

6 Click the **Solve** button on the Main toolbar.

7 Open the **Solver Parameters** dialog box again, and select **Time dependent** from the **Solver** list.

8 In the **Times** edit field type `linspace(0,2,101)`, and in the **Absolute tolerance** edit field type `Az 1e-3 lm1 5e3 omega 0.015`. This sets different tolerances for each variable.

9 Click **OK**.

10 Open the **Solver Manager**, click the **Solve For** tab, and select all variables.

11 Click **OK**.

12 Click the **Restart** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 Open the **Plot Parameters** dialog box. From the **Solution at time** list on the **General** page, select the solution for time 0.2 s.

2 On the **Contour** page, select **Magnetic potential, z component** from the **Predefined quantities** list and type `15` in the **Levels** edit field. Select the **Contour plot** check box.

3 In the **Contour color** area, click the **Uniform color** button. Click the **Color** button. Select a black color, then click **OK** to close the **Contour Color** dialog box. Clear the **Color scale** check box.

4 Click **OK** to close the **Plot Parameters** dialog box.

You should now see the following surface plot of the norm of the magnetic flux with contours indicating magnetic flux field lines.

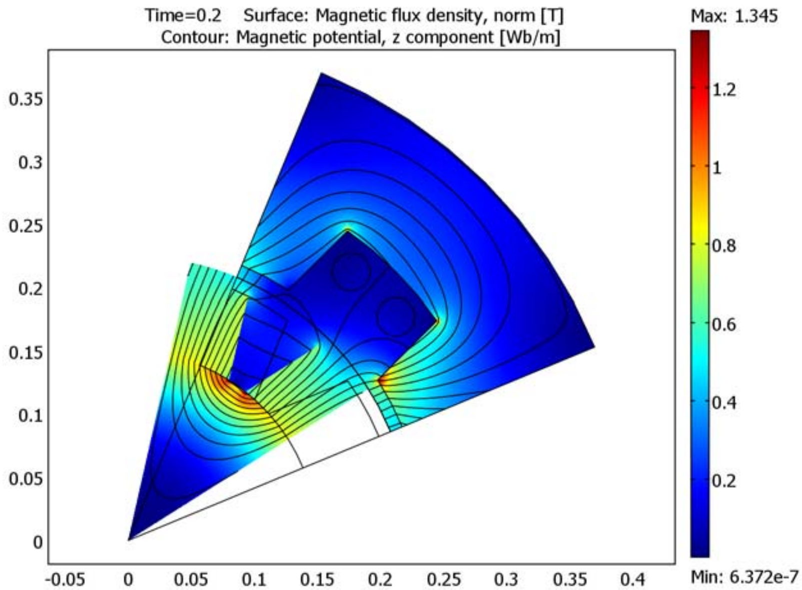


Figure 3-8: Magnetic flux norm (surface) and field lines (contours) at t=0.2 s.

To view the rotation angle as a function of time, proceed with the steps below.

1 From the **Postprocessing** menu, select **Global Variables Plot**.

2 In the **Predefined quantities** area, choose **Rotation angle**, then click the **>** button.

3 Click **Apply** to generate Figure 3-7 on page 55.

Using the same dialog box, create plots for `omegat`, `Tz`, `(M+Tz)/(I+I0)`, and `Vi/Rc` to plot angular velocity, electromagnetic torque, acceleration, and winding current, respectively, by repeating the steps given below for the angular velocity.

1 Select the item in the **Quantities to plot** frame and click the **<** button to remove it from the list.

2 In the **Expression** edit field enter `omegat`.

3 Click the **>** button to the right of the edit field.

4 Click **Apply** to view the plot.

To create an animation of the rotating rotor, perform the following steps:

1 Open the **Plot Parameters** dialog box, click the **Animate** tab, and make sure that all time steps are selected.
2 Click the **Start Animation** button. When the movie has been generated, the COMSOL Movie Player opens it.

Generator in 3D

Introduction

This model is a 3D version of the 2D generator model on page 39, so most of the details about the model are explained there. The main difference is that this is a static example, calculating the magnetic fields around and inside the generator.

Modeling in COMSOL Multiphysics

The model has some differences compared to the 2D generator model. The PDE is simplified to solve for the magnetic scalar potential, V_m, instead of the vector potential, $\mathbf{A}$. It is based on the assumption that currents can be neglected, which holds true when the generator terminals are open. The equation for V_m becomes

$$-\nabla \cdot (\mu \nabla V_m - \mathbf{B}_\mathrm{r}) = 0$$

The stator and center of the rotor are made of annealed medium-carbon steel (soft iron), which is a nonlinear magnetic material. This is implemented in COMSOL Multiphysics as an interpolation function of the B-H curve of the material. The difference compared to the 2D version is that the norm of the magnet flux, $|\mathbf{B}|$, has to be calculated from the norm of the magnetic field, $|\mathbf{H}|$, in this model.

Results and Discussion

Figure 3-9 shows the norm of the magnetic flux for a slice through a centered cross section of the generator. The plot also shows the streamlines of the magnetic flux. The starting points of the streamlines have been carefully selected to show the closed loops between neighboring stator and rotor poles. A few streamlines are also plotted at the edge of the generator to illustrate the field there.

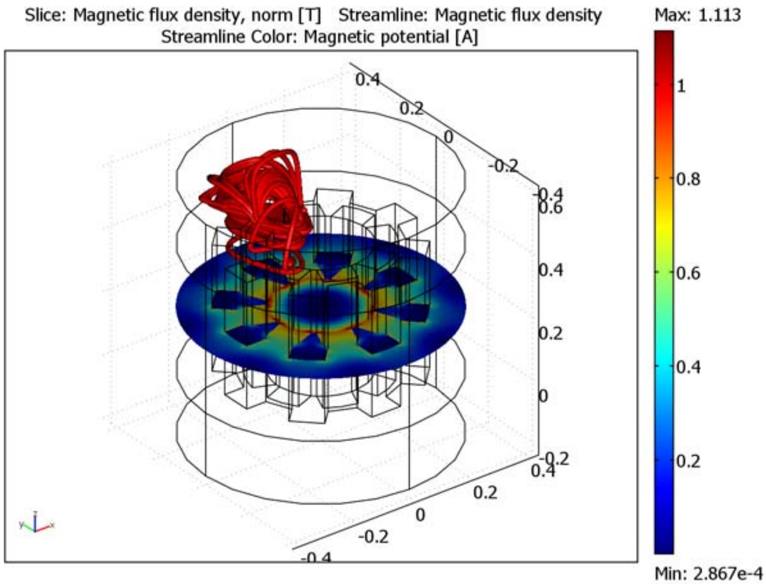


Figure 3-9: A combined slice and streamline plot of the magnetic flux density.

Model Library path: `ACDC_Module/Motors_and_Drives/generator_3d`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **3D** in the **Space dimension** list.

2 In the **AC/DC Module** folder, select **Statics>Magnetostatics, No Currents**.

3 Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

1 Go to **Draw>Work-Plane Settings**.

2 In the dialog box that appears, click **OK**.

3 Go to **Draw>Specify Objects>Circle** to create circles with the following properties:

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
C1	0.3	Center	0	0	22.5
C1	0.235	Center	0	0	0

4 Go to **Draw>Specify Objects>Rectangle** to create rectangles with the following properties:

NAME	WIDTH	HEIGHT	BASE	X	Y	ROTATION ANGLE
R1	0.1	1	Center	0	0	0
R2	0.1	1	Center	0	0	45
R3	0.1	1	Center	0	0	90
R4	0.1	1	Center	0	0	135

5 In the **Create Composite Object** dialog box, clear the **Keep interior boundaries** check box.

6 Enter the formula `C2+C1*(R1+R2+R3+R4)`.

7 Click **OK**.

8 Create a circle with the following properties:

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
C1	0.215	Center	0	0	0

9 Draw four rectangles with the following properties:

NAME	WIDTH	HEIGHT	BASE	X	Y	ROTATION ANGLE
R1	0.1	1	Center	0	0	22.5
R2	0.1	1	Center	0	0	-22.5
R3	0.1	1	Center	0	0	67.5
R4	0.1	1	Center	0	0	-67.5

10 In the **Create Composite Object** dialog box, enter the formula `C1*(R1+R2+R3+R4)`.

11 Click **OK**.

12 Create a circle with the following properties:

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
C1	0.15	Center	0	0	22.5

13 In the **Create Composite Object** dialog box, make sure that the **Keep interior boundaries** check box is not selected.

14 Enter the formula `C1+CO2`.

15 Click **OK**.

16 Draw a circle with parameters according to the table below.

NAME	RADIUS	BASE	X	Y	ROTATION ANGLE
C1	0.15	Center	0	0	22.5

17 Click the **Zoom Extents** button to get a better view of the geometry.

18 Go to **Draw>Extrude**.

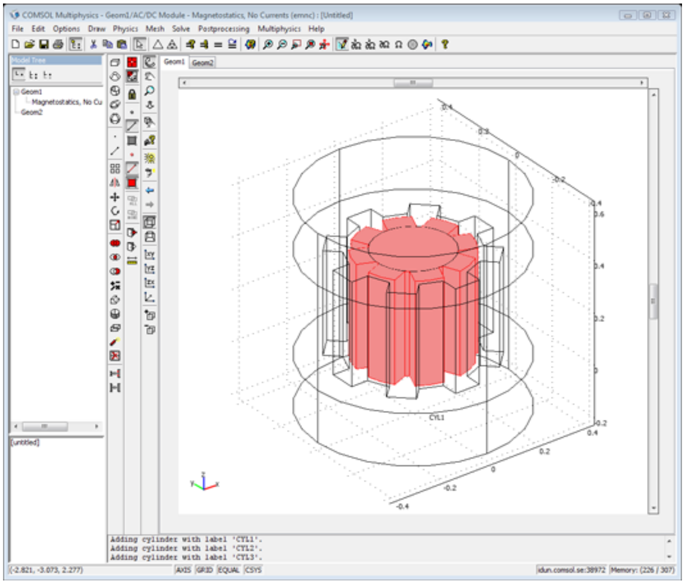
19 In the dialog box that appears, select all objects and type `0.4` in the **Distance** edit field.

20 Click **OK**.

21 Use the **Cylinder** tool to create cylinders with parameters according to the table below.

NAME	AXIS BASE POINT Z	RADIUS	HEIGHT
CYL1	-0.2	0.4	0.2
CYL2	0	0.4	0.4
CYL3	0.4	0.4	0.2

22 Click the **Zoom Extents** button to get a better view of the geometry.



PHYSICS SETTINGS

Constants

1 From the **Options** menu, select **Constants**.

2 In the **Constants** dialog box, define the following constants with names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
BrSmCo	0.84[T]	Remanent flux in permanent magnets
murSmCo	1	Relative permeability in permanent magnets

Scalar Expressions

1 From the **Options** menu, select **Expressions>Scalar Expressions**.

2 In the **Scalar Expressions** dialog box, define the following expression; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
R	sqrt(x^2+y^2)	Radial coordinate

Subdomain Settings

1 Enter subdomain settings for the active subdomains according to the table below.

SETTINGS	SUBDOMAINS 1, 3, 4	SUBDOMAINS 5, 9, 10, 13	SUBDOMAINS 6, 8, 11, 12	SUBDOMAINS 2, 7
Library material				Soft Iron (without losses)
$\mathbf{B} \leftrightarrow \mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$	$\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$	$\mathbf{B} = f(\lvert\mathbf{H}\rvert)\mathbf{e}_H$
μ_r	1	`murSmCo`	`muSmCo`	
B_{rx}		`-BrSmCo*x/R`	`BrSmCo*x/R`	
B_{ry}		`-BrSmCo*y/R`	`BrSmCo*y/R`	

For Subdomains 2 and 7, choose the material from the **Electric (AC/DC) Material Properties** library in the **Materials/Coefficients Library** dialog box, which you open by clicking **Load**.

2 Click **OK**.

Boundary Conditions

In the **Boundary Settings** dialog box, apply the **Magnetic insulation** condition to all boundaries, then click **OK**.

MESH GENERATION

1 In the **Free Mesh Parameters** dialog box, select **Finer** from the **Predefined mesh sizes** list.

2 Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

1 Open the **Plot Parameters** dialog box.

2 On the **General** page, select the check boxes for **Slice** plot and **Streamline** plot.

3 Click the **Slice** tab. From the **Predefined quantities** list, select **Magnetic flux density, norm**.

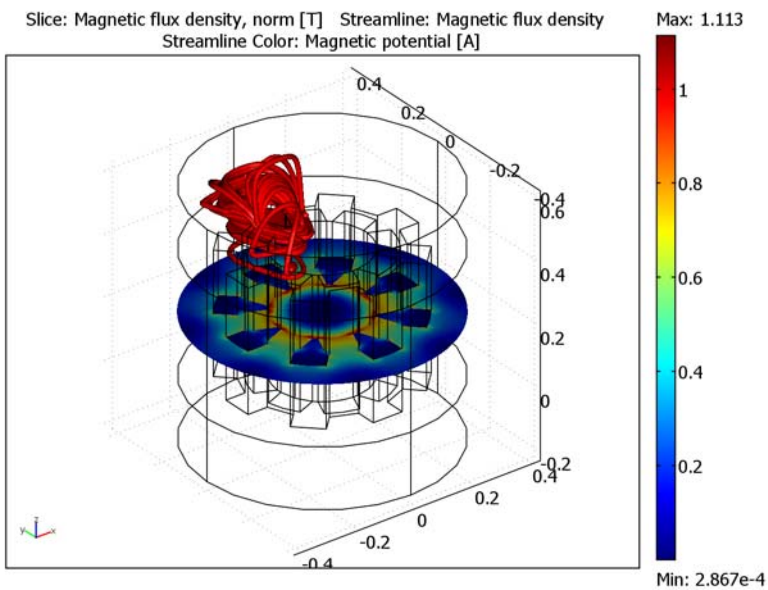
4 Type `0` in the edit field for **x levels**, and type `1` in the edit field for **z levels**.

5 Click the **Streamline** tab and select **Magnetic flux density** in the **Predefined quantities** list.

6 Click the **Specify start point coordinates** button and enter the values according to the table below.

FIELD	EXPRESSION
x	`0 0 0 0 0 0 0`
y	`0.35 0.35 0.35 0.35 0.35 0.3 0.2`
z	`linspace(0.1,0.59,7)`

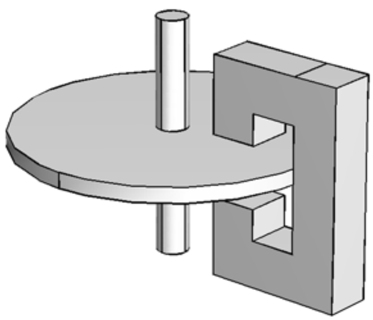
7 Select **Tube** in the **Line type** list, then click **OK**.



Magnetic Brake in 3D

Introduction

A magnet brake in its simplest form consists of a disk of conductive material and a permanent magnet. The magnet generates a constant magnetic field, in which the disk is rotating. When a conductor moves in a magnetic field it induces currents, and the Lorentz forces from the currents slow the disk.



Model Definition

This 3D problem is solved using a stationary formulation for the electromagnetic part coupled to an ordinary differential equation for the rotational rigid body dynamics. It also illustrates the proper use of a Lorentz type induced current density term. An ungauged A-V formulation is used to solve the electromagnetic part in a fast and memory efficient way.

For a disk rotating with angular velocity ω about the z-axis, the velocity $\mathbf{v}$ at a point (x, y) is given by

$$\mathbf{v} = \omega(-y, x, 0)$$

Maxwell-Ampères law, expressed using a magnetic vector potential $\mathbf{A}$, a scalar electric potential V, and an induced current density term of Lorentz type is the fundamental field equation used in this model. It is complemented by a current balance but no explicit gauging is provided.

$$\nabla \times (\mu^{-1}\nabla \times \mathbf{A}) - \sigma\mathbf{v} \times (\nabla \times \mathbf{A}) + \sigma\nabla V = 0$$
$$-\nabla \cdot (-\sigma\mathbf{v} \times (\nabla \times \mathbf{A}) + \sigma\nabla V) = 0$$

Such a formulation is inherently singular because the divergence of the magnetic vector potential is not uniquely determined and there is an infinite number of possible solutions to the problem. All such solutions yield the same magnetic flux and current densities, however. It is further known that the above formulation, when solved using an iterative solver, converges quickly and robustly provided that any explicitly supplied source current density (zero here) is divergence free and no direct coarse grid solver is involved in the solution scheme.

The induced Lorentz current density term is a common source of confusion in electromagnetic modeling. In situations when the moving domain is of bounded extent in the direction of the motion or varies in this direction or contains magnetic sources that also move, Lorentz terms cannot be used. This is because the part of the magnetic flux generated by moving sources must not be included in the Lorentz term. In this situation, the induced current distribution is stationary (it stays where the magnet is and does not move with the spinning disk). Thus, the moving domain does not contain any magnetic sources that move along with it and it is unbounded and invariant in the direction of the motion.

The magnetic and electric boundary conditions on external boundaries are

$$\mathbf{n} \times \mathbf{A} = 0, \qquad \mathbf{n} \cdot \mathbf{J} = 0$$

Now consider how the system evolves over time. The induced torque slows the disk down, described by an ordinary differential equation (ODE) for the angular velocity ω.

$$\frac{d\omega}{dt} = \frac{T_z}{I},$$

Where the torque T_z is obtained as the z component of the vector

$$\mathbf{T} = \int_{\text{disk}} \mathbf{r} \times (\mathbf{J} \times \mathbf{B})dV$$

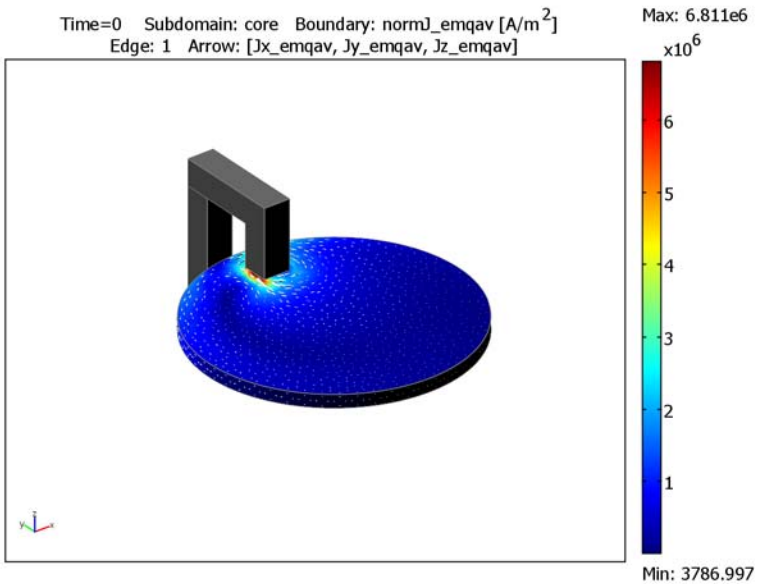
and the moment of inertia I for a disk with radius r and unit thickness equals

$$I = m\frac{r^2}{2} = \frac{\rho r^4 \pi}{2}$$

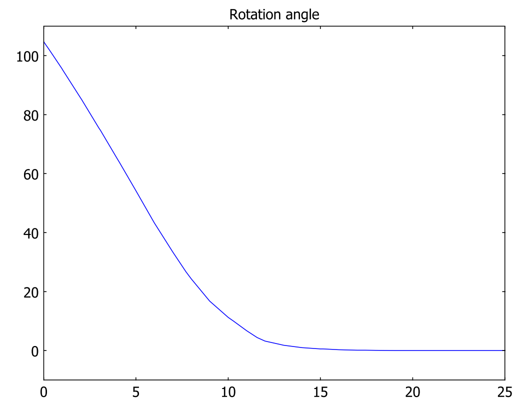
Here m is the disk mass and ρ is the density.

Results and Discussion

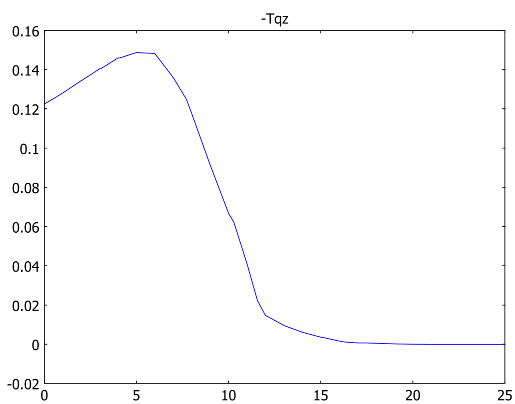
The COMSOL Multiphysics model is set up for a 1 cm thick copper disk with a radius of 10 cm and an initial angular speed of 1000 rpm. The magnet consists of a 1 T, hard ($\mu_r = 1$) permanent magnet connected to an iron yoke with a 1.5 cm air gap in which the copper disk spins. The figures below show the induced eddy current density and the time evolution of the angular velocity, braking torque and dissipated power repectively.



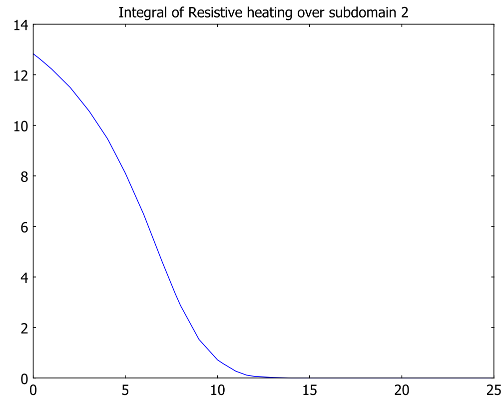
The eddy current magnitude and direction at t = 0 s.



The time evolution of the angular velocity.



The time evolution of the braking torque.



The time evolution of the dissipated power.

Model Library path: `ACDC_Module/Motors_and_Drives/magnetic_brake_3d`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **3D** from the **Space dimension** list.
2 In the **AC/DC Module** folder, select **Statics>Magnetostatics, Ungauged AV**. This node contains an application mode that solves for magnetic (A) and electric (V) potentials using solver settings for a special ungauged technique.
3 Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

1 Use the **Cylinder** tool to create a cylinder with **Radius** `0.1`, **Height** `0.01` and **Axis base point z:** `-0.005`. All other parameters are kept at their defaults.

   Use the **Sphere** tool to create a sphere with a **Radius** of `0.3` and other parameters at their defaults.
2 Go to **Draw>Work-Plane Settings**.
3 In the dialog box that appears, select the **y-z** plane at $x = 0$. Click **OK**.

4 Go to **Draw>Specify Objects>Rectangle** to create rectangles with the following properties:

NAME	WIDTH	HEIGHT	BASE	X0	Y0
R1	`0.02`	`-0.0075+0.06`	Corner	`0.06`	`-0.06`
R2	`0.06`	`0.02`	Corner	`0.08`	`-0.06`

5 From the **Draw** menu, open the **Create Composite Object** dialog box.

6 Clear the **Keep interior boundaries** check box.

7 Enter the formula `R1+R2` in the **Set formula** edit field.

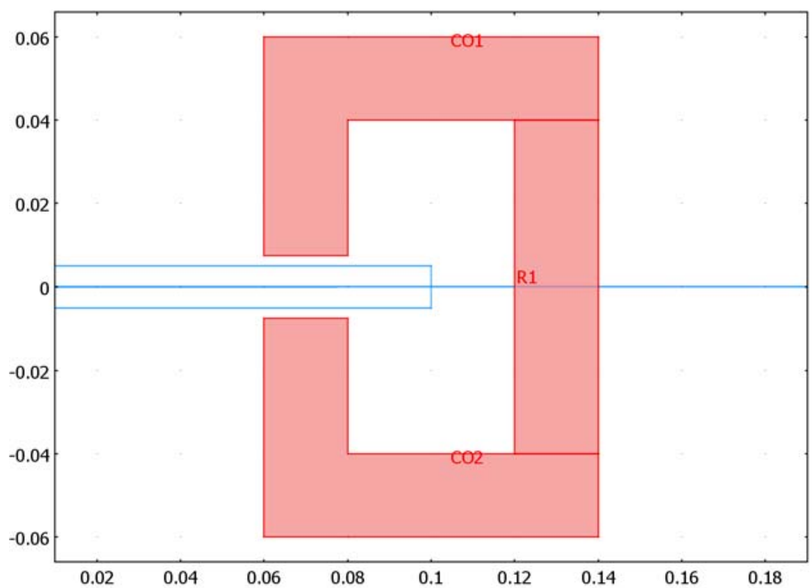
8 Click **OK**.

9 With the newly formed composite object selected, click the **Mirror** button, change the **Normal vector** to `(0,1)` and click **OK**.

10 Create another rectangle with the following properties:

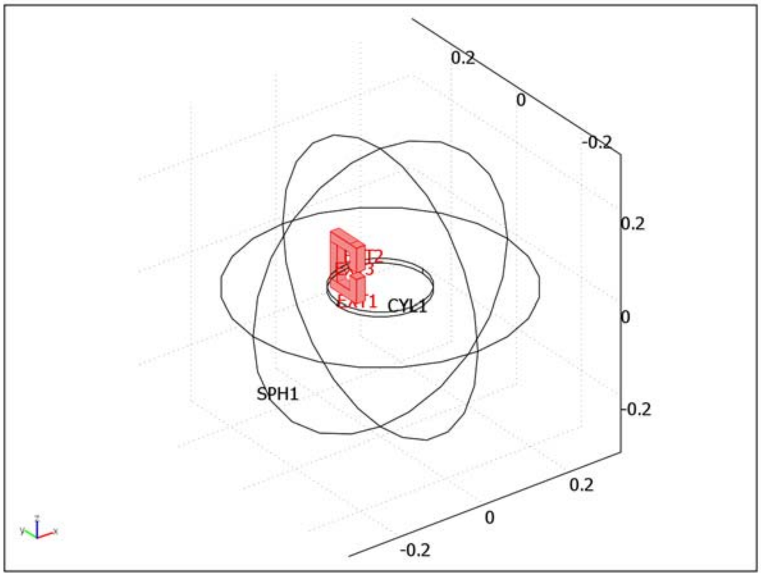
NAME	WIDTH	HEIGHT	BASE	X0	Y0
R1	`0.02`	`0.08`	Corner	`0.12`	`-0.04`

11 You now have three objects, which together form the cross section of the magnet.



12 Select all three objects and select **Extrude** from the **Draw** menu. Extrude a distance of `0.02`.

13 In the 3D view, with the extrusion objects selected, click the **Move** toolbar button and move the objects `-0.01` in the *x* direction.



OPTIONS AND SETTINGS

Constants

1 From the **Options** menu, choose **Constants**.

2 Enter the following constants; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`rpm`	`1000[1/min]`	`Initial rpm value`
`W0`	`2*pi*rpm`	`Initial angular velocity`
`I0`	`0[kg*m^2]`	`External inertia`

Expression Variables

1 On the **Options** menu, point to **Expressions** and then click **Scalar Expressions**.

2 Enter Lorentz force variables according to the following table; when done, click **OK**.

NAME	EXPRESSION
Fx	Jy_emqav*Bz_emqav-Jz_emqav*By_emqav
Fy	Jz_emqav*Bx_emqav-Jx_emqav*Bz_emqav
Fz	Jx_emqav*By_emqav-Jy_emqav*Bx_emqav

3 On the **Options** menu, point to **Expressions** and then click **Subdomain Expressions**.

4 Enter the following subdomain expression; when done, click **OK**.

NAME	SUBDOMAINS 3–5
core	1

Coupling Variables

Define integration coupling variables for the torque and the moment of inertia.

1 On the **Options** menu, point to **Integration Coupling Variables** and then click **Subdomain Variables**.

2 Enter the following variables; when done, click **OK**.

NAME	SUBDOMAIN 2
Iz	8700*(x^2+y^2)
Tqz	x*Fy-y*Fx

PHYSICS SETTINGS

Subdomain Settings

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

2 Enter subdomain settings for the active subdomains according to the table below. To change the constitutive relation, you need to choose the relation from the **Constitutive relation** list. Click the **Load** button to access the material libraries. For subdomain 2, you can use the predefined material **Copper** that you find in the **Basic Material Properties** material library.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAINS 3,4	SUBDOMAIN 5
Material		Copper		
Velocity x	0	-y*W	0	0
Velocity y	0	x*W	0	0
Velocity z	0	0	0	0

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAINS 3,4	SUBDOMAIN 5
Electric conductivity	1	5.998e7[S/m]	1	1
Constitutive relation	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$
Rel. permeability	1	1	4000	1
Rem. flux density x				0
Rem. flux density y				0
Rem. flux density z				1

Boundary Conditions

Use the default magnetic boundary condition but change the electric boundary condition to **Electric insulation**.

Space-Independent Equations

1 From the **Physics** menu, open the **Space-Independent Equations** dialog box and enter entities according to the following table. The description field is optional.

NAME	EQUATION	INIT(U)	INIT(UT)	DESCRIPTION
`W`	`Wt-Tqz/(Iz+I0)`	`W0`	`0`	`Rotation angle`

Make sure that **SI** is selected in the **Base unit system** list.

2 Click **OK**.

MESH GENERATION

1 Go to Subdomain Mode by clicking the **Subdomain Mode** toolbar button.

2 Select Subdomain 2 (copper disk) only.

3 Open the **Free Mesh Parameters** dialog box by selecting **Free Mesh Parameters** from the **Mesh** menu.

4 Select **Extremely fine** from the list of **Predefined mesh sizes**.

5 Click the **Mesh Selected** button to mesh just the disk.

6 Select **Coarser** from the list of **Predefined mesh sizes**.

7 Click **OK**.

8 Click the **Mesh Remaining (Free)** button on the Mesh toolbar to mesh the rest of the geometry.

COMPUTING THE SOLUTION

1 Open the **Solver Parameters** dialog box by clicking the corresponding button on the Main toolbar or select it from the **Solve** menu.

2 Set **Solver** to **Time dependent**.

3 In the **Times** edit field type `0:1:25`.

4 In the **Relative tolerance** edit field type `0.001`.

5 In the **Absolute tolerance** edit field type `W 0.1 V 1e-5 tAxAyAz10 1e-7 tAxAyAz20 1e-7 tAxAyAz21 1e-7`.

6 Click the **Time Stepping** tab and set **Time steps taken by solver** to **Intermediate**.

7 Click **OK** to close the **Solver Parameters** dialog box.

8 Open the **Probe Plot Parameters** dialog box from the **Postprocessing** menu.

9 Click **New** and select **Global** from the **Plot type** list.

10 Type `Omega` for **Plot name**.

11 Click **OK**.

12 Type `W` in the **Expression** edit field.

13 Click **New** and select the **Global** from the **Plot type** list.

14 Type `Torque` for **Plot name**.

15 Click **OK**.

16 Type `-Tqz` in the **Expression** edit field.

17 Click **New** and select **Integration** as **Plot type** and **Subdomain** as **Domain type**.

18 Type `Power` for **Plot name**.

19 Click **OK**.

20 Type `Q_emqav` in the **Expression** edit field and select Subdomain 2 from the **Subdomain selection** list.

21 Click **OK**.

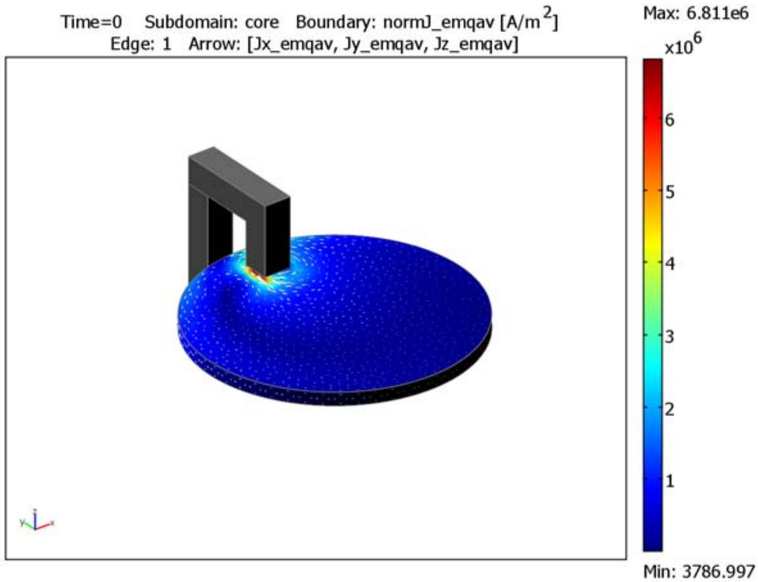
22 Click the **Solve** button on the Main toolbar to compute the solution. The model takes of the order of 10–20 minutes and uses less than 1 GB of memory to solve on a standard PC. The probe plots already shown in the results section appear as the solution is in progress.

POSTPROCESSING AND VISUALIZATION

1 Open the **Plot Parameters** dialog box from the **Postprocessing** menu.

2 Clear the check box for **Slice** plot and select the ones for **Subdomain**, **Boundary**, **Edge**, and **Arrow** plots on the **General** page. Select **0** from the **Solution at time** list.
3 Click the **Subdomain** tab. Type `core` in the **Expression** field. Set the **Element color** to **Uniform color**. Click the **Color** button and choose a medium gray color from the palette.
4 Click the **Boundary** tab. Type `normJ_emqav` in the **Expression** field.
5 Click the **Edge** tab. Type `1` in the **Expression** field. Select **gray** from the **Colormap** list and clear the **Color scale** check box.
6 Click the **Arrow** tab. Select **Boundaries** from the **Plot arrows on** list. Click the **Boundary Data** tab and enter `Jx_emqav`, `Jy_emqav`, and `Jz_emqav`, respectively, in the edit fields. Set the **Arrow type** to **3D arrow**. Clear the **Auto** check box for **Scale factor** and enter `2` in the associated edit field.
7 Move the **Plot Parameters** dialog box to the side but keep it open.
8 From the **Options** menu, select **Suppress>Suppress Subdomains**.
9 Select Subdomains 1 and 2.
10 Click **OK**.
11 From the **Options** menu, select **Suppress>Suppress Boundaries**.
12 Select all boundaries except numbers 5–8, 32, and 35.
13 Click **OK**.
14 From the **Options** menu, select **Suppress>Suppress Edges**.
15 Select Edges 1–5, 21, 31, 44–47, 50–54, 57, and 76.
16 Click **OK**.
17 Click **OK** in the **Plot Parameters** dialog box.
18 Click the **Scene Light** button on the Camera toolbar.

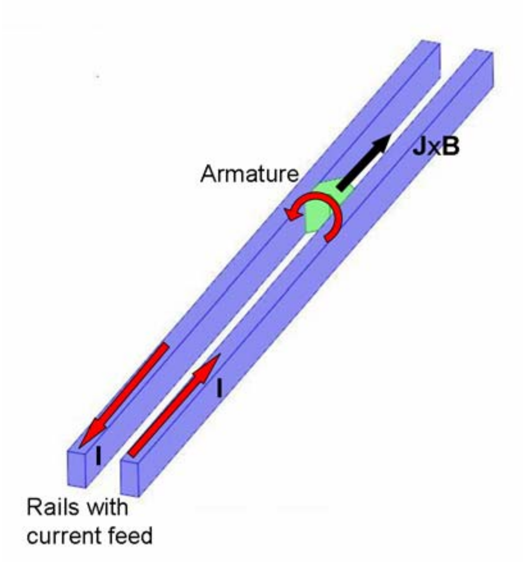
19 In the status bar at the bottom of the main window, double-click **AXIS**.



Railgun

Introduction

A railgun is a type of magnetic accelerator gun that utilizes an electromagnetic force to propel an electrically conductive projectile (armature) that is initially part of the current path. The current flowing through the rails sets up a magnetic field between them and through the projectile perpendicular to the current in it. This results in an acceleration of the projectile along the rails. As no ferromagnetic materials are involved, there are no saturation effects and the muzzle speed of the projectile is only limited by the available electric power and the structural and thermal durability of the device. In theory, muzzle speeds of several km/s are possible.



Model Definition

This 3D problem is solved using a transient inductive formulation for the electromagnetic part coupled to an ordinary differential equation for the lumped kinematics of the armature. The railgun is a predominantly inductive and resistive device where the input power is transformed into three different entities:

- Joule heating

- Magnetic energy as more and more of the rails become energized as the armature moves forward
- Kinetic energy of the armature

As the current is limited by inductive and resistive effects, capacitive effects do not play any major role in the dynamics of the railgun itself. A practical railgun implementation may be driven by a high energy capacitor bank that can be included as a driving circuit (not implemented here).

Maxwell-Ampères law, expressed using a magnetic vector potential $\mathbf{A}$ and a scalar electric potential V, is the fundamental field equation used in this model. It is complemented by a DC current balance.

$$\sigma\frac{\partial \mathbf{A}}{\partial t} + \nabla \times (\mu^{-1}\nabla \times \mathbf{A}) = -\sigma\nabla V$$
$$\nabla \cdot (-(\sigma\nabla V)) = 0$$

The armature is not modeled as a separate domain but as a moving conductivity distribution that is interpolated on a fixed mesh. Note that induced current terms of Lorentz type cannot be used in this modeling situation as the armature is of finite extent in the direction of motion and it carries an externally driven flux source that moves along with the motion. That is, part of the magnetic flux is stationary in the moving frame, see also the discussion in connection with the model "Magnetic Brake in 3D" on page 72.

The magnetic and electric boundary conditions on external boundaries are;

$$\mathbf{n} \times \mathbf{A} = 0$$
$$\mathbf{n} \cdot \mathbf{J} = 0$$

except for the voltage driven ends of the rails where suitable constraints are applied on the electric potential.

Now consider how the system evolves over time. The Lorentz force accelerates the armature. This is modeled as an ordinary differential equation (ODE) for the armature position x_0.

$$\frac{dx_0}{dt} = \frac{F_x}{\mathrm{M}}$$

Where M is the armature mass and the force F_x is obtained as the x-component of the volume integral of the Lorentz force density over the armature:

$$\mathbf{F} = \int_V (\mathbf{J} \times \mathbf{B}) dV$$

Results and Discussion

The COMSOL Multiphysics model is set up for an armature that is a 4 cm cube made of aluminum. Each rail has a cross section that is 4 cm by 2 cm and a length of 4.8 m. As the armature is initially inserted 4 cm, the effective length is slightly shorter. The rails are made of copper. The device is driven by a constant applied voltage of 300 V. The following figures contain snapshots of the solution and time traces of rail current, position, speed, and the accelerating Lorentz force.

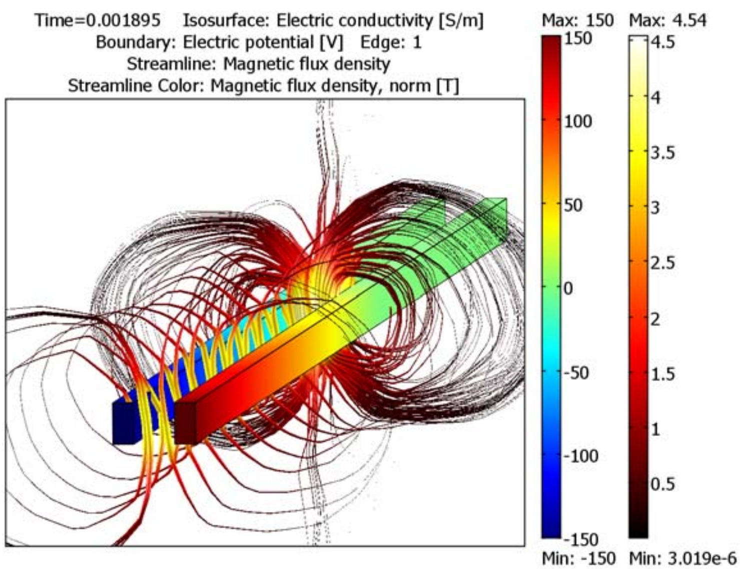
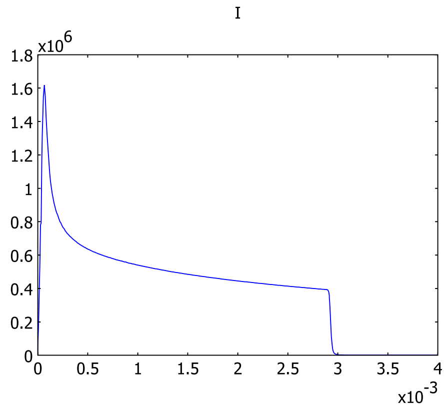
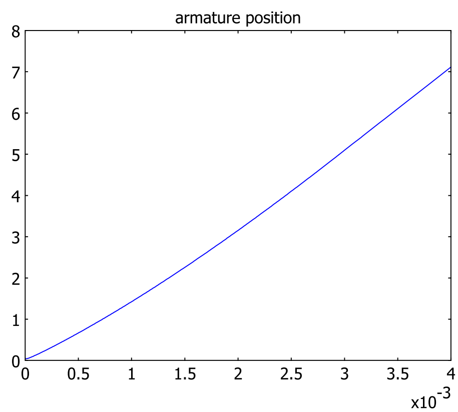


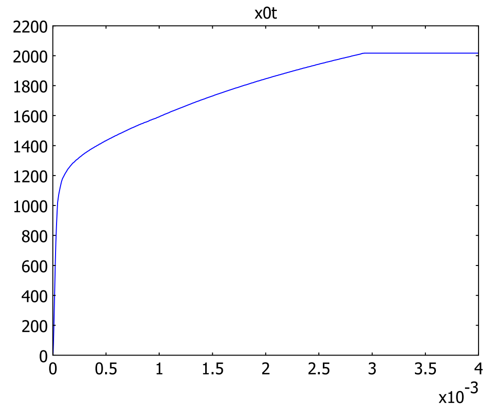
Figure 3-10: A snapshot of the solution at t = 1.895 ms. The electric potential is plotted as color on the rails. Streamlines and streamline color represent the magnetic flux density and magnitude respectively.



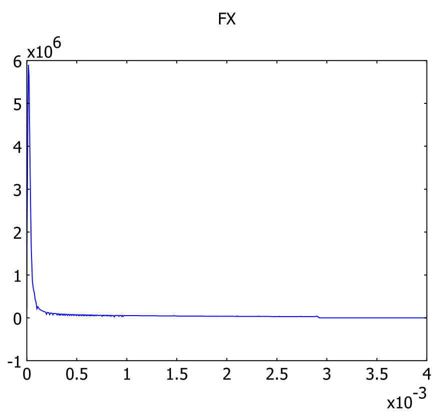
The time evolution of the rail current.



The time evolution of the armature position.



The time evolution of the armature velocity.



The time evolution of the Lorentz force is shown.

Model Library path: `ACDC_Module/Motors_and_Drives/railgun_3d`

MODEL NAVIGATOR

1 Select **3D** from the **Space dimension** list.

2 Click the **Multiphysics** button.

3 In the **AC/DC Module** folder, select **Quasi-Statics, Magnetic>Induction Currents>Transient analysis**.

4 Select **Vector - Linear** from the **Element** list.

5 Click **Add** to include the selected application mode in the list to the right.

6 In the **AC/DC Module** folder, select **Statics>Conductive Media DC**.

7 Select **Lagrange - Linear** in the **Element** list.

8 Click **Add** to include the selected application mode in the list to the right.

9 Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

1 Go to **Draw>Work-Plane Settings**.

2 In the dialog box that appears, select the **y-z plane** at $x = 0$, click **OK**.

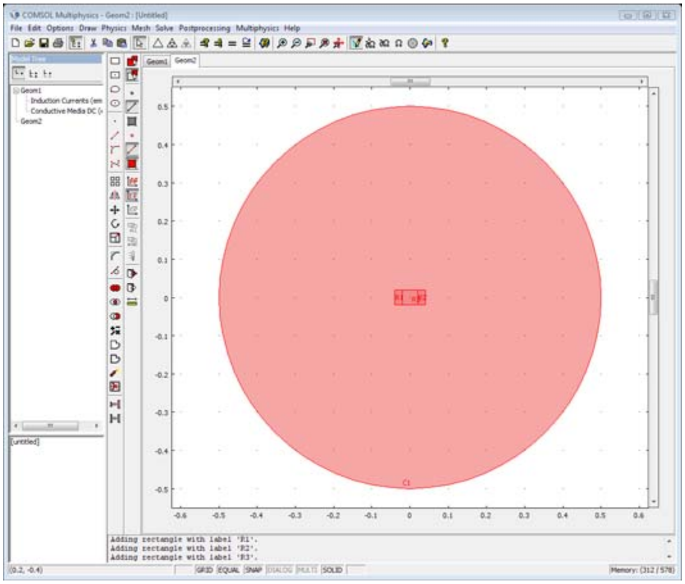
3 Go to **Draw>Specify Objects>Circle** to create a circle with a radius of 0.5 and the center at (0, 0).

4 Go to **Draw>Specify Objects>Rectangle** to create rectangles with the following properties:

NAME	WIDTH	HEIGHT	BASE	X0	Y0
R1	0.02	0.04	Corner	-0.04	-0.02
R2	0.02	0.04	Corner	0.02	-0.02
R3	0.08	0.04	Corner	-0.04	-0.02

5 Select all objects and click the **Zoom Extents** button on the Main toolbar.

6 You will now have three objects that together form the cross section of the railgun.



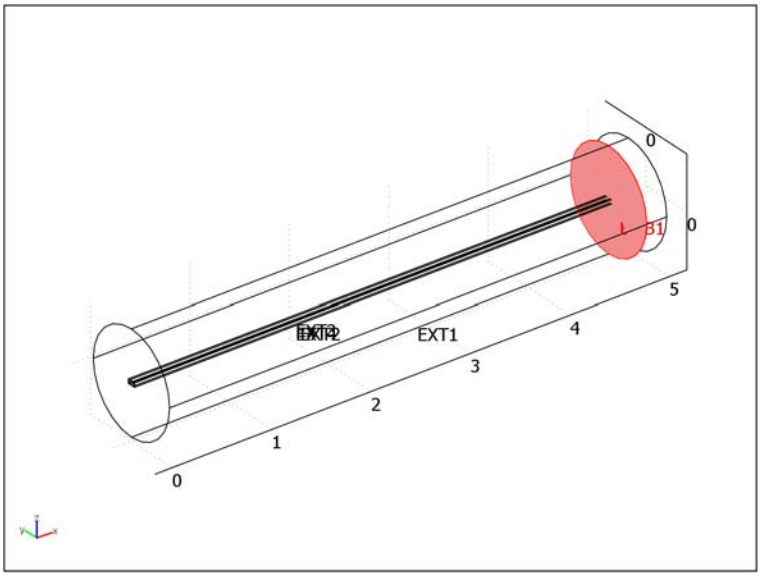
7 Select all four objects and select **Extrude** from the **Draw** menu. Extrude a distance of 5.

8 Click on the **Geom2** tab in the main window to go back to the 2D geometry.

9 Select only the circle object and select **Embed** from the **Draw** menu.

10 Click **OK**.

11 In the 3D geometry with the embedded circle object selected, click the **Move** toolbar button and move it a distance of 4.8 in the *x* direction.



OPTIONS AND SETTINGS

Constants

1 From the **Options** menu, choose **Constants**.

2 Enter the following constants; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`V`	`0.04^3[m^3]`	`Armature volume`
`rho_Al`	`2700[kg/m^3]`	`Armature density`
`M`	`rho_Al*V`	`Armature mass`
`w`	`0.04[m]`	`Armature length`
`k`	`100`	`Constant in exponential factor`
`sigma_Al`	`1/2.65e-8[ohm*m]`	`Armature conductivity`
`V0`	`300[V]`	`Applied voltage`

Expression Variables

1 On the **Options** menu, point to **Expressions** and then click **Scalar Expressions**.

2 Enter Lorentz force variables according to:

NAME	EXPRESSION	DESCRIPTION
`Fx`	`Jy_emqa*Bz_emqa-Jz_emqa*By_emqa`	`Lorentz force x`
`Fy`	`Jz_emqa*Bx_emqa-Jx_emqa*Bz_emqa`	`Lorentz force y`
`Fz`	`Jx_emqa*By_emqa-Jy_emqa*Bx_emqa`	`Lorentz force z`

Coupling Variables

Define integration coupling variables for the force on the armature and the rail current.

1 On the **Options** menu, point to **Integration Coupling Variables** and then click **Subdomain Variables**.

2 Enter the following variable; when done, click **OK**.

NAME	SUBDOMAIN 3
`FX`	`Fx`

3 On the **Options** menu, point to **Integration Coupling Variables** and then click **Boundary Variables**.

4 Enter the following variable:

NAME	BOUNDARIES 1, 4
`I`	`nJ_emdc-Jix_emqa`

5 Click **OK**.

PHYSICS SETTINGS

Subdomain Settings

1 In the **Multiphysics** menu, make sure that the **Conductive Media DC** application mode is selected.

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

3 Select Subdomains 1 and 5 and deselect the **Active in this domain** box to avoid solving this part of the problem in the air.

4 Enter subdomain settings for the active subdomains according to the table below.

SETTINGS	SUBDOMAINS 2, 4	SUBDOMAIN 3	SUBDOMAINS 6–8
Electric conductivity	`5.98e7`	`1+sigma_Al*exp(-k*(abs(x-x0)>(w/2))*(abs(x-x0)-(w/2)))`	`1`

Boundary Conditions

Use the **Electric insulation** boundary condition for all active boundaries except for Boundaries 4 and 14 where you apply an **Electric potential** of `V0/2` and `-V0/2`, respectively.

Subdomain Settings

1 From the **Multiphysics** menu, select the **Induction Currents** application mode.

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

3 Enter subdomain settings for the active subdomains according to the table below.

SETTINGS	SUBDOMAINS 1, 5	SUBDOMAINS 2, 4	SUBDOMAIN 3	SUBDOMAINS 6–8
Electric conductivity	1	5.98e7	1+sigma_Al*exp(-k*(abs(x-x0)>(w/2))*(abs(x-x0)-(w/2)))	1
J^e_x	0	Jx_emdc	Jx_emdc	Jx_emdc
J^e_y	0	Jy_emdc	Jy_emdc	Jy_emdc
J^e_z	0	Jz_emdc	Jz_emdc	Jz_emdc

Boundary Conditions

Use the default **Magnetic insulation** boundary condition for all active boundaries.

Space-Independent Equations

1 From the **Physics** menu, open the **Space-Independent Equations** dialog box and enter entities according to the following table. The description field is optional.

NAME	EQUATION	INIT(U)	INIT(UT)	DESCRIPTION
x0	x0tt-FX/M	0.04	0	armature position

2 Click **OK**.

MESH GENERATION

1 Go to Boundary Mode by clicking on the **Boundary Mode** button on the Main toolbar.

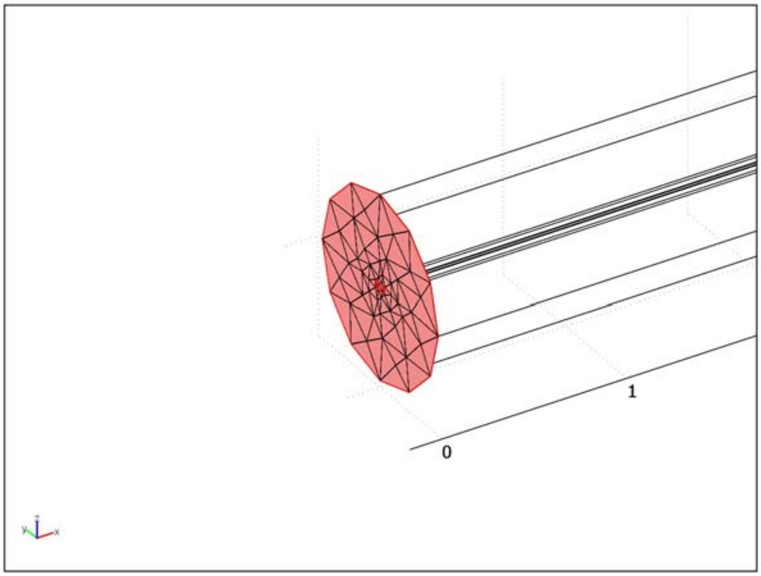
2 Select Boundaries 1, 4, 8, and 14 (the railgun inlet end).

3 Open the **Free Mesh Parameters** dialog box by selecting **Free Mesh Parameters** from the **Mesh** menu.

4 Select **Coarse** from the list of **Predefined mesh sizes**.

5 Click the **Mesh Selected** button to mesh the inlet end of the railgun.

6 Click **OK**.



7 Go to Subdomain Mode by clicking on the **Subdomain Mode** button on the Main toolbar.

8 Open the **Swept Mesh Parameters** dialog box by selecting **Swept Mesh Parameters** from the **Mesh** menu.

9 Select Subdomains 1–4 in the **Subdomain selection** list.

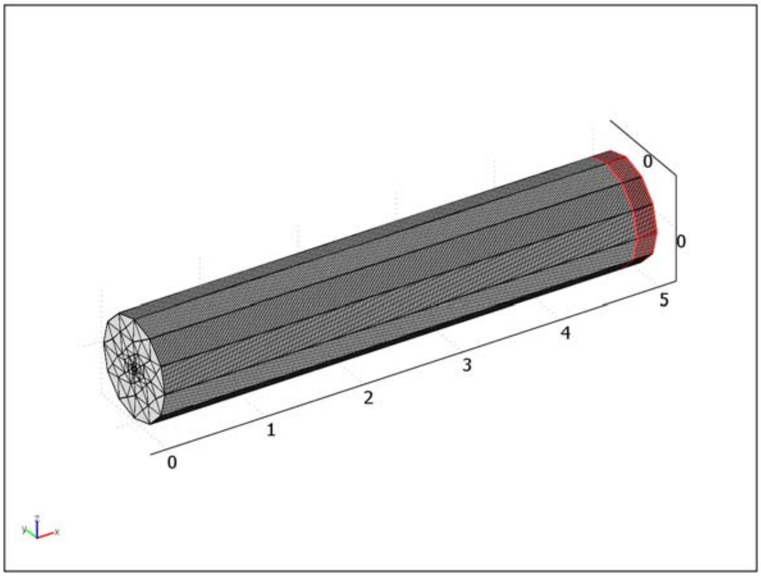
10 Select the **Manual specification of element layers** check box and specify `240` for **Number of element layers**.

11 Click the **Mesh Selected** button.

12 Select Subdomains 5–8 in the **Subdomain selection** list.

13 Select the **Manual specification of element layers** check box and specify `10` for **Number of element layers**.

14 Click the **Mesh Selected** button.



15 Click **OK**.

COMPUTING THE SOLUTION

1. Open the **Solver Parameters** dialog box by clicking the corresponding button on the Main toolbar or select it from the **Solve** menu.
2. Set **Solver** to **Time dependent**.
3. In the **Times** edit field type `linspace(0,4e-3,20)`.
4. In the **Absolute tolerance** edit field type: `tAxAyAz10 0.03e-4 V2 0.03 x0 0.01`.
5. Set the **Linear system solver** to **Direct (PARDISO)**.
6. Click **OK** to close the **Solver Parameters** dialog box.
7. Open the **Probe Plot Parameters** dialog box from the **Postprocessing** menu.
8. Click **New** and select **Global** from the **Plot type** list.
9. Type `Position` in the **Plot name** edit field.
10. Click **OK**.
11. Type `x0` in the **Expression** field.
12. Click **New** and select **Global** from the **Plot type** list.

13 Type `Velocity` for **Plot name**.

14 Click **OK**.

15 Type `x0t` in the **Expression** field.

16 Click **New** and select **Global** from the **Plot type** list.

17 Type `Current` for **Plot name**.

18 Click **OK**.

19 Type `I` in the **Expression** field.

20 Click **New** and select **Global** from the **Plot type** list.

21 Type `Force` for **Plot name**.

22 Click **OK**.

23 Type `FX` in the **Expression** field.

24 Click **OK**.

25 Click the **Solve** button on the Main toolbar to compute the solution.

The model takes on the order of 2–4 hours and uses less than 1 GB of memory to solve on a standard PC. The probe plots already shown in the results section appears as the solution is in progress.

POSTPROCESSING AND VISUALIZATION

To reproduce Figure 3-10, follow these instructions:

1 From the **Options** menu, select **Suppress>Suppress Subdomains**.

2 Select Subdomains 2, 4, 6, and 8.

3 Click **OK**.

4 From the **Options** menu, select **Suppress>Suppress Boundaries**.

5 Select Boundaries 1–3, 8, 10–13, 19–21, 26, 28–31, 37, and 39.

6 Click **OK**.

7 From the **Options** menu, select **Suppress>Suppress Edges**.

8 Select Edges 1–3, 10, 12, 14–17, 26–31, 33, 35, 36, 38, 40–45, 47, 49, 52–54, and 59–62.

9 Click **OK**.

10 Open the **Plot Parameters** dialog box from the **Postprocessing** menu.

11 Clear the check box for **Plot type**: **Slice** and **Geometry edges**, and select the ones for **Isosurface**, **Boundary**, **Edge** and **Streamline** on the **General** page.

12 On the **General** page, select the **Solution at time** 0.001895.

13 Click the **Isosurface** tab. Type sigma_emqa in the **Expression** field. Set the **Isosurface levels** to **Vector with isolevels** and enter 1e7 in the corresponding field. Set **Isosurface color** to **Uniform color**. Click the **Color** button and choose a yellow color from the palette. Clear the **Color scale** check box.

14 Click the **Boundary** tab. Type V2 in the **Expression** field.

15 Click the **Edge** tab. Type 1 in the **Expression** field. Set **Edge color** to **Uniform color**. Click the **Color** button and choose black from the palette.

16 Click the **Streamline** tab.

17 Set the **Streamline data** fields to Bx_emqa, By_emqa, and Bz_emqa, respectively.

18 Click the **Start Points** tab, select **Specify start point coordinates** and enter [linspace(0.02,5,20) linspace(0.02,5,20)], [0.01*ones(1,20) -0.01*ones(1,20)], 0 in the **x**, **y**, and **z** data fields, respectively.

19 Click the **Line Color** tab, select **Use expression** and click the **Color Expression** button.

20 Type normB_emqa in the **Expression** field.

21 Select **hot** from the **Colormap** list.

22 Click **OK** to close the **Streamline Color Expression** dialog box.

23 Set the **Line type** to **Tube** and click the **Tube Radius** button.

24 Select the **Radius data** check box.

25 Type normB_emqa in the **Expression** field.

26 Clear the **Auto** check box for **Radius scale factor** and enter 0.1.

27 Click **OK** to close the **Tube Radius Parameters** dialog box.

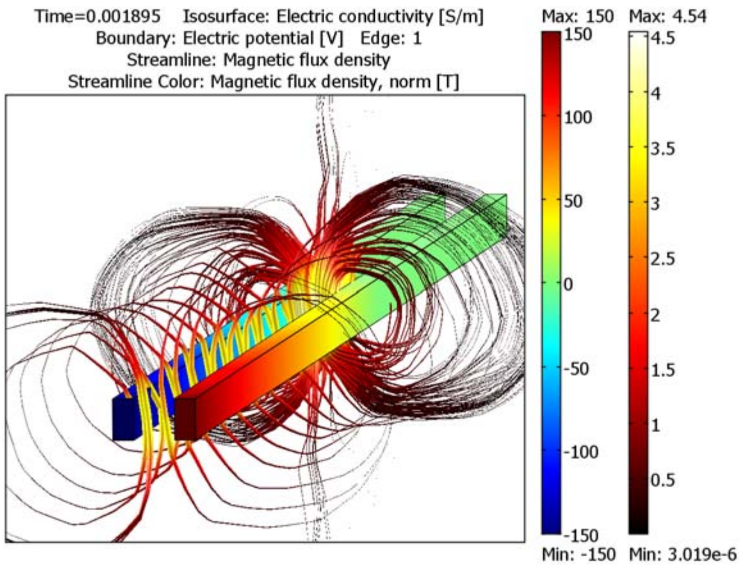
28 Click the **Advanced** button and decrease the **Maximum number of integration steps** to 1000.

29 Click **OK** to close the **Advanced Streamline Parameters** dialog box.

30 Click **OK** in the **Plot Parameters** dialog box.

31 Click the **Scene Light** button on the Camera toolbar.

32 In the status bar at the bottom of the main window, double-click **AXIS** and **CSYS**. Use the mouse to adjust the view.



4

Electrical Component Models

This chapter contains models of electrical components. In most cases, the models make use of the static and quasi-static application modes in the AC/DC Module.

Tunable MEMS Capacitor

Introduction

In an electrostatically tunable parallel plate capacitor you can modify the distance between the two plates when the applied voltage changes. For tuning of the distance between the plates the capacitor includes a spring that attaches to one of the plates. If you know the characteristics of the spring and the voltage between the plates, you can compute the distance between the plates. This model includes an electrostatic simulation for a given distance. A postprocessing step then computes the capacitance.

The capacitor in this model is a typical component in various microelectromechanical systems (MEMS) for electromagnetic fields in the radio frequency range 300 MHz to 300 GHz.

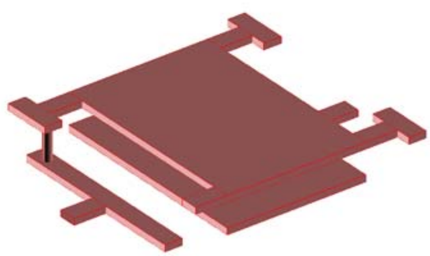


Figure 4-1: The tunable MEMS capacitor consists of two metal plates. The distance between the plates is tuned via a spring connected to one of the plates.

Model Definition

To solve the problem, use the 3D Electrostatics application mode in the AC/DC Module. The capacitance is available directly as a variable for postprocessing.

DOMAIN EQUATIONS

The electric scalar potential, V, satisfies Poisson's equation,

$$-\nabla\cdot(\varepsilon_0\varepsilon_r\nabla V) = \rho$$

where ε_0 is the permittivity of free space, ε_r is the relative permittivity, and ρ is the space charge density. The electric field and the displacement are obtained from the gradient of V:

$$\mathbf{E} = -\nabla V$$
$$\mathbf{D} = \varepsilon_0 \varepsilon_r \mathbf{E}$$

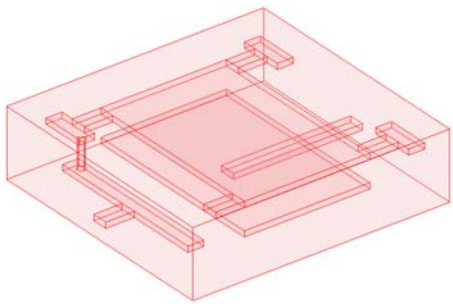


Figure 4-2: The computational domain. The interiors of the capacitor plates and the connecting bars are not included.

BOUNDARY CONDITIONS

Potential boundary conditions are applied to the capacitor plates and bars. A port condition maintains the potential 1 V at the upper plate and the connecting bars, whereas the lower plate is kept at ground potential. For the surface of the surrounding box, apply conditions corresponding to zero surface charge at the boundary,

$$\mathbf{n} \cdot \mathbf{D} = 0$$

Results and Discussion

Figure 4-3 shows the computed electric field and potential distribution in the capacitor. The potential on each capacitor plate is constant, as dictated by the boundary condition. Right between the plates, the electric field is almost uniform, but close to the edges of the plates, you can see fringing fields that change magnitude and direction rapidly in space.

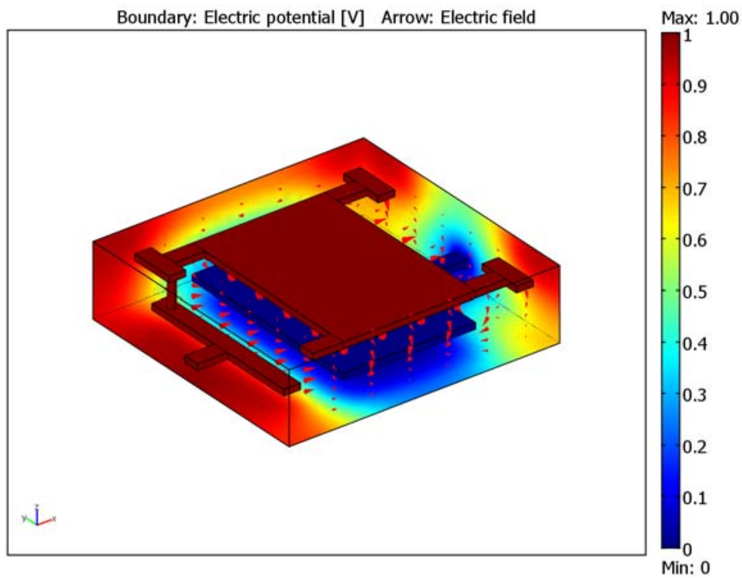


Figure 4-3: The electric potential is shown as a surface plot while the cones indicate the strength and orientation of the electric field.

The capacitance, C, obtained from the simulation is approximately 0.09 pF.

Model Library path: `ACDC_Module/Electrical_Components/capacitor`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **3D** in the **Space dimension** list.

2 In the list of application modes, select **AC/DC Module>Statics>Electrostatics**.

3 Click **OK**.

OPTIONS AND SETTINGS

In the **Axes/Grid Settings** dialog box clear the **Axis auto** check box and specify axis settings according to the following table.

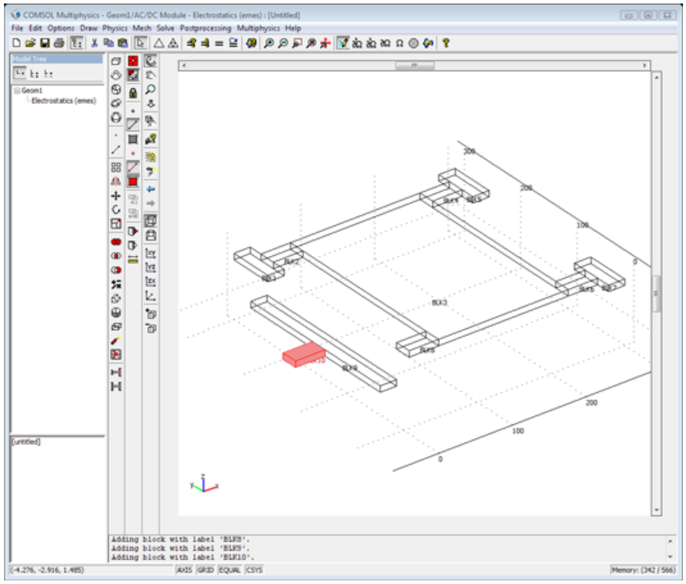
AXIS	MIN	MAX
x	-50	300
y	-50	300
z	-25	75

GEOMETRY MODELING

The geometry of this model can be created either by drawing in a work plane and then extruding the 2D geometry or by using 3D primitives. The advantage of the latter approach is that the coordinates of the blocks can be given explicitly.

1 Draw solid blocks according to the following table.

NAME	LENGTH			AXIS BASE POINT		
	X	Y	Z	X	Y	Z
BLK1	22	60	8	0	240	46
BLK2	40	22	8	22	259	46
BLK3	176	262	8	62	19	46
BLK4	40	22	8	238	259	46
BKL5	22	60	8	278	240	46
BLK6	40	22	8	238	19	46
BLK7	22	60	8	278	0	46
BLK8	40	22	8	22	19	46
BLK9	22	229	8	0	41	0
BLK10	40	22	8	-40	139	0



2 Draw a solid cylinder with radius 5.5 and height 38 with the axis parallel to the z-axis, and the base point of the axis located at (11, 250, 8).

3 Select all geometry objects using the **Select All** command from the **Edit** menu, then open the **Create Composite Object** dialog box. Be sure to clear the **Keep interior boundaries** check box and verify that all objects are selected. Then click **OK** to create a composite object as the union of all objects. The advantage with removing the interior boundaries in this geometry is that it makes it easier to create a high quality mesh later on.

4 Now, draw two more solid block objects according to the following table.

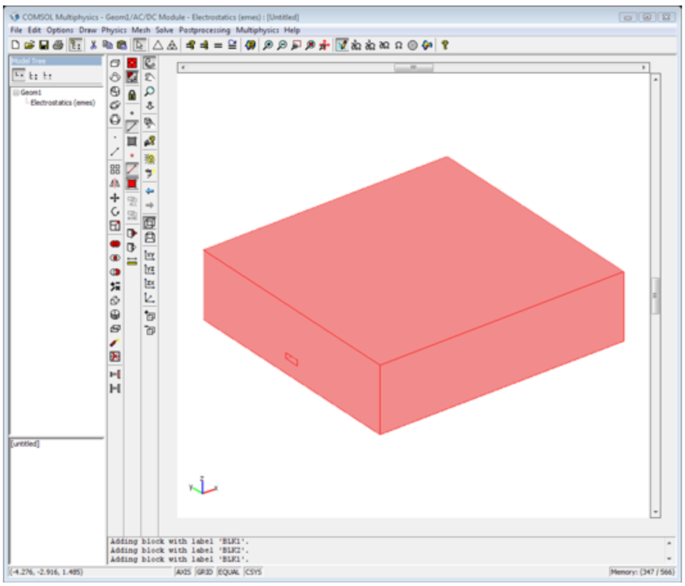
NAME	LENGTH			AXIS BASE POINT		
	X	Y	Z	X	Y	Z
BLK1	176	262	8	62	19	8
BLK2	181	22	8	139	139	0

5 Using the Shift key, select both block objects, then click the **Union** button to form another composite object. The interior boundaries of this object are not kept because the settings in the **Create Composite Object** dialog box are applied for all set operations until modified again.

6 Now draw a block surrounding the two objects corresponding to the two plates in the capacitor. Create a block of size 360-by-340-by-94, with the base point of the axis at (–40, –20, –20).

7 Once again open the **Create Composite Object** dialog box, and now enter the formula `BLK1-(CO1+CO2)` in the **Set formula** edit field.

8 The geometry of the model is now ready, except for the size. The capacitor dimensions you have entered should be given in µm. This is easily accounted for by clicking the **Scale** button and giving the value `1e-6` for all three scaling factors. Then click the **Zoom Extents** button on the Main toolbar to see the object.



PHYSICS SETTINGS

Subdomain Settings

Because there is silica-glass between the plates, you need to modify the permittivity. Enter the following values for the subdomain settings:

SETTINGS	SUBDOMAIN 1
ε_r	4.2
ρ	0

Boundary Conditions

The lower plate is grounded while a positive potential is applied to the upper plate. The surrounding boundaries are kept electrically insulated. The following table shows the corresponding boundary settings.

SETTINGS	BOUNDARIES 37–40, 45, 47–51, 57	BOUNDARIES 1–4, 9, 76	ALL OTHERS
Boundary condition	Ground	Zero charge/Symmetry	Port
Port tab			Use port as input
Input property			Energy method

MESH GENERATION

1 Open the **Free Mesh Parameters** dialog box. On the **Global** tab, select **Coarse** from the **Predefined mesh sizes** list. Then select **Custom mesh size** and set the **Resolution of narrow regions** parameter to `1.5` to help the mesh generator resolve the thin regions in the geometry.

2 Click **Remesh** to initialize the mesh, then click **OK** to close the **Free Mesh Parameters** dialog box.

COMPUTING THE SOLUTION

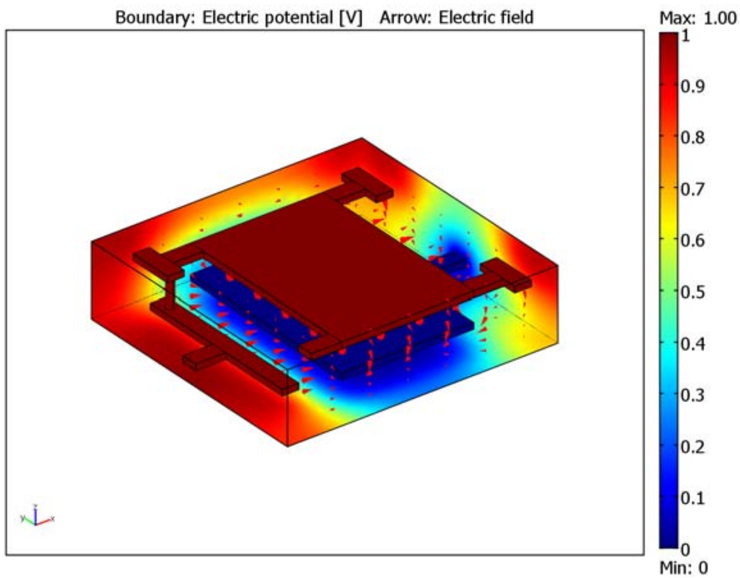
Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

Use the electric field distribution together with a plot of the potential to visualize the result of the simulation. You have access to the capacitance as a variable, `C11_emes`, that you can plot or display directly.

1 Start by suppressing some boundaries not to be used for visualization. Select **Suppress Boundaries** from the **Options** menu. Select Boundaries 1, 2, and 4, then click **OK**.

2 In the **Plot Parameters** dialog box on the **General** tab clear **Slice**, then select **Boundary** and **Arrow** as **Plot type**. Use the default settings for both plot types.



3 Choose **Postprocessing>Data Display>Global** to open the **Global Data Display** dialog box. Type `C11_emes` in the **Expression** edit field, then click **OK** to display the capacitance in the message log.

Modeling Using the Programming Language

```
% Start the modeling by clearing the fem variable in
% the COMSOL Script or MATLAB workspace.
clear fem

% Begin by specifying space dimensions and defining a geometry.
fem.sdim = {'x','y','z'};

blks{1} = block3(22,60,8,'corner',[0 240 46]);
blks{2} = block3(40,22,8,'corner',[22 259 46]);
blks{3} = block3(176,262,8,'corner',[62 19 46]);
blks{4} = block3(40,22,8,'corner',[238 259 46]);
blks{5} = block3(22,60,8,'corner',[278 240 46]);
blks{6} = block3(40,22,8,'corner',[238 19 46]);
blks{7} = block3(22,60,8,'corner',[278 0 46]);
blks{8} = block3(40,22,8,'corner',[22 19 46]);
```

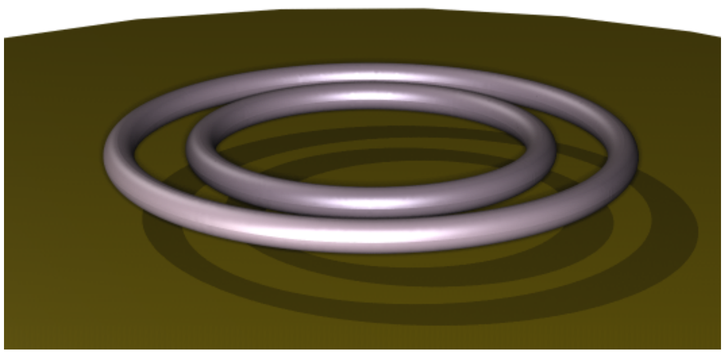
```
blks{9} = block3(22,218,8,'corner',[0 41 0]);
blks{10} = block3(40,22,8,'corner',[-40 139 0]);
cyl1 = cylinder3(5.5,38,[11 250 8]);
co1 = geomcomp({blks{:} cyl1},'face','all','edge','all');

blk1 = block3(176,262,8,'corner',[62 19 8]);
blk2 = block3(181,22,8,'corner',[139 139 0]);
co2 = blk1 + blk2;

blk3 = block3(360,340,94,'corner',[-40 -20 -20]);
co3 = geomcomp({co1 co2 blk3},'ns',{'CO1','CO2','BLK1'}, ...
  'sf','BLK1-(CO1 + CO2)','face','all','edge','all');

fem.geom = scale(co3,1e-6,1e-6,1e-6,0,0,0);

% The application mode structure contains all the physical
% properties of the model. Create this with the following
% commands:
clear appl

appl.mode = 'EmElectrostatics';

appl.equ.rho0 = '0';
appl.equ.epsilonr = '4.2';

% Use Energy method for capacitance calculation
clear prop
prop.input='We';
appl.prop = prop;

% Assign zero charge, port, and ground boundary conditions to
% correct boundary numbers.
appl.bnd.inport = {0,1,0};
appl.bnd.type = {'nD0','port','V0'};
appl.bnd.ind = ones(1,76)*2;
appl.bnd.ind(1,[1:4 9 76]) = 1;
appl.bnd.ind(1,[37:40 45 47:51 57]) = 3;

% Generate the coefficients in the FEM structure.
fem.appl = appl;
fem = multiphysics(fem);

% Create a coarse mesh for the geometry with the following commands.
% The extended mesh structure contains all the necessary data
% defining the different elements.
fem.mesh = meshinit(fem, ...
  'hauto',6, ...
  'hnarrow',1.5);

fem.xmesh = meshextend(fem);
```

```
% Solve the problem and visualize the result.
fem.sol = femstatic(fem);

% Plot the voltage on all boundaries except on number 1, 2, and 4.
% The electric field is plotted as arrows for all subdomains.
bdlList = setdiff(1:76,[1 2 4]);
postplot(fem,...
  'bdl', bdlList, ...
  'tridata',{'V','cont','on'}, ...
  'tribar', 'on', ...
  'arrowdata',{'Ex','Ey','Ez'}, ...
  'axisequal','on', ...
  'axisvisible','off')

% The capacitance value (evaluated at an arbitrary point):
I1 = postinterp(fem,'C11',[0;0;0])
```

Induction Currents from Circular Coils

Introduction

A time-varying current induces a varying magnetic field. This field induces currents in neighboring conductors. The induced currents are called eddy currents. The following model illustrates this phenomenon by a time-harmonic field simulation as well as a transient analysis, which provides a study of the eddy currents resulting from switching on the source.



Two current-carrying coils are placed above a copper plate. They are surrounded by air, and there is a small air gap between the coils and the metal plate. A potential difference provides the external source. To obtain the total current density in the coils you must take the induced currents into account. The time-harmonic case shows the skin effect, that is, that the current density is high close to the surface and decreases rapidly inside the conductor.

Model Definition

EQUATION

To solve the problem in the AC/DC Module, use a quasi-static equation for the magnetic potential **A**.

$$\sigma\frac{\partial\mathbf{A}}{\partial t} + \nabla\times(\mu_0^{-1}\mu_r^{-1}\nabla\times\mathbf{A}) = \sigma\frac{V_{\mathrm{loop}}}{2\pi r}$$

where μ_0 is the permeability of vacuum, μ_r the relative permeability, σ the electric conductivity, and V_{loop} is the voltage over one turn in the coil. In the time-harmonic case the equation reduces to

$$j\omega\sigma\mathbf{A} + \nabla \times (\mu_0^{-1}\mu_r^{-1}\nabla \times \mathbf{A}) = \sigma\frac{V_{\text{loop}}}{2\pi r}$$

FORCES

It is also possible to compute the forces on the plate caused by the eddy currents. The following expression gives the Lorentz force on the plate:

$$\mathbf{F} = \mathbf{J} \times \mathbf{B}$$

When dealing with time-harmonic fields, you must take the complex nature of the field representation into special consideration. In the time domain, the force in the axial direction in this axisymmetric model is

$$F_z = -J_\varphi B_r = -\text{Re}(J'_\varphi e^{j\omega t}) \cdot \text{Re}(B'_r e^{j\omega t})$$

where J'_φ and B'_r are the current and magnetic flux in the frequency domain, obtained from the harmonic equation.

Results and Discussion

The applied voltage is constant across each wire, but the induced current makes the current density higher toward the end facing the copper plate. Figure 4-4 below shows this and also displays the field lines from the magnetic flux.

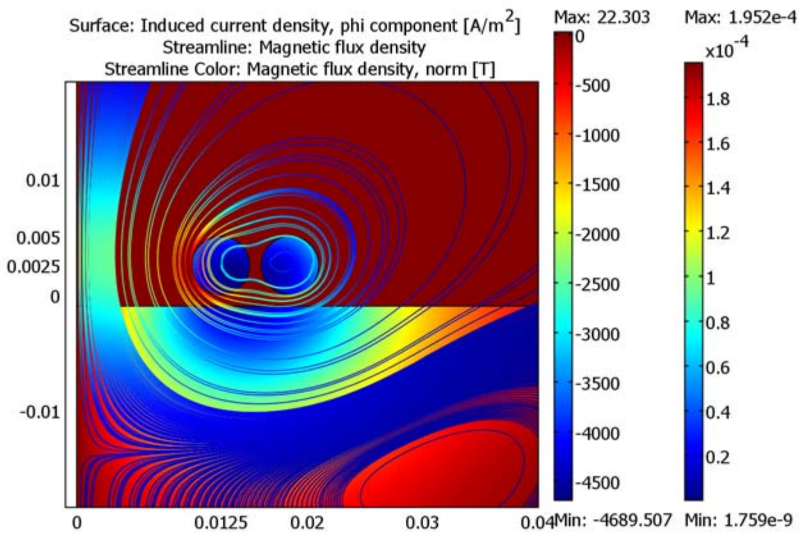


Figure 4-4: The φ component of the current density plotted together with the field lines of the magnetic flux.

The eddy currents in the wires and in the copper plate give rise to a force between the parts. Figure 4-5 below shows a plot of the force on the plate over one period.

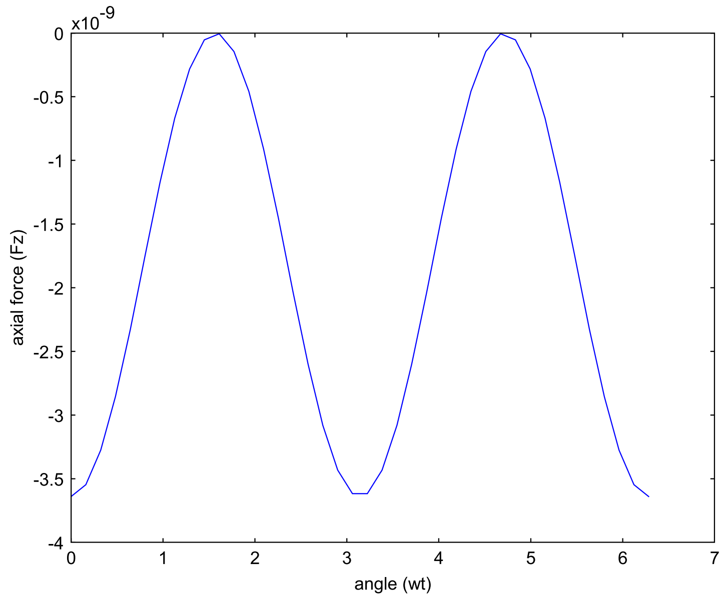


Figure 4-5: The force on the copper plate over one period. Although the solution only gives the amplitude and phase, these can easily be converted to a real signal in the time domain.

Model Library path: `ACDC_Module/Electrical_Components/coil_above_plate`

Harmonic Analysis in the Graphical User Interface

MODEL NAVIGATOR

1 From the **Space dimension** list, select **Axial symmetry (2D)**.

2 Select the **AC/DC Module>Quasi-Statics, Magnetic>Azimuthal Induction Currents, Vector Potential>Time-harmonic analysis** application mode.

3 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Axes/Grid Settings**.

2 Set axis and grid settings according to the following table.

AXIS		GRID	
r min	-0.05	r spacing	0.01
r max	0.15	Extra r	0.0125
z min	-0.08	z spacing	0.01
z max	0.08	Extra z	0.0025 0.005

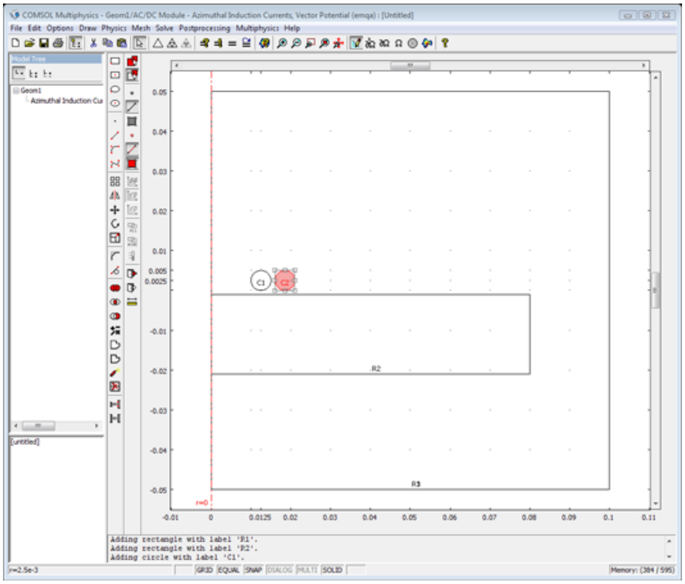
GEOMETRY MODELING

1 Draw the surrounding rectangle R1 with opposite corners at (0, –0.05) and (0.1, 0.05).

2 Draw the rectangle representing the metal plate R2 with the corners at (0, –0.02) and (0.08, 0). Now change the position of R2 by pressing the **Move** button on the Draw toolbar or selecting it in the **Draw** menu under **Modify**. Set the **z displacement** to –0.001.

3 Draw a circle C1 representing the first current-carrying coil, centered at (0.0125, 0.0025) with radius 0.0025.

4 Select C1 and click the **Array** button. Set the **Displacement** in the *r* direction to `0.006` and the **Array size** in the *r* direction to `2`.



PHYSICS SETTINGS

Scalar Variables

From the **Physics** menu, select **Scalar Variables** and set the frequency variable `nu_emqa` to `50`.

Subdomain Settings

In this model, use the COMSOL Multiphysics materials library to specify material parameters in the subdomains. Only the conductivity, the permeability, and the permittivity are determined by the material selected.

1 From the **Physics** menu, open the **Subdomain Settings** dialog box, and select Subdomains 2–4.

2 Under the **Electric Parameters** tab, click the **Load** button. In the dialog box that appears, select **Copper** and click **OK**.

3 Enter subdomain settings according to the following table.

SETTING	SUBDOMAINS 1, 2	SUBDOMAINS 3, 4
V_{loop}	0	1e-4

Boundary Conditions

Set boundary conditions according to the following table.

SETTING	BOUNDARIES 1, 3, 5	BOUNDARIES 2, 7, 9
Boundary condition	Axial symmetry	Magnetic insulation

MESH GENERATION

1 Initialize the mesh by clicking the **Initialize Mesh** button on the Main toolbar.

2 Click the **Refine Mesh** button to refine the mesh.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The magnetic potential is the default visualization quantity. In this case it is of interest to plot the eddy currents together with the magnetic flux density. These fields are used to examine the magnetic forces that arise in current-carrying conductors.

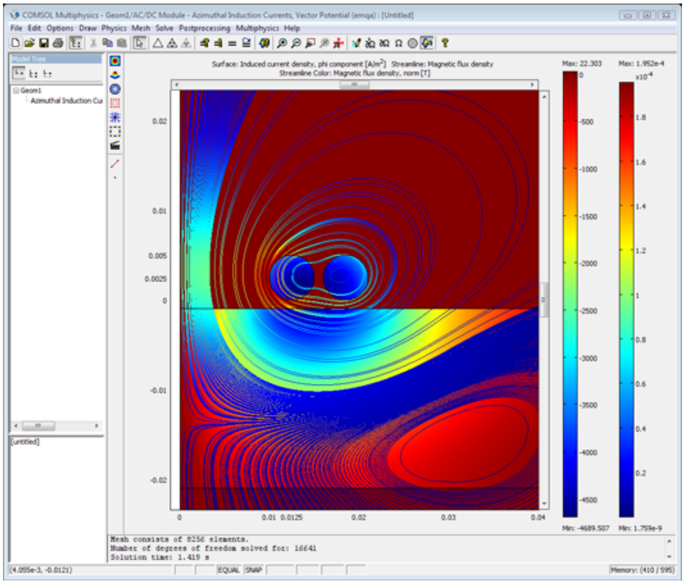
1 Click the **Plot Parameters** button on the Main toolbar.

2 On the **General** tab select **Surface** plot and **Streamline** plot.

3 On the **Surface** tab select **Induced current density, phi component** as **Surface data**.

4 On the **Streamline** tab select **Magnetic flux density** as **Streamline data**.

5 For a better view of the current-intensive region, use the **Zoom Window** button on the Main toolbar. The current density is highest near the edge of the plate.

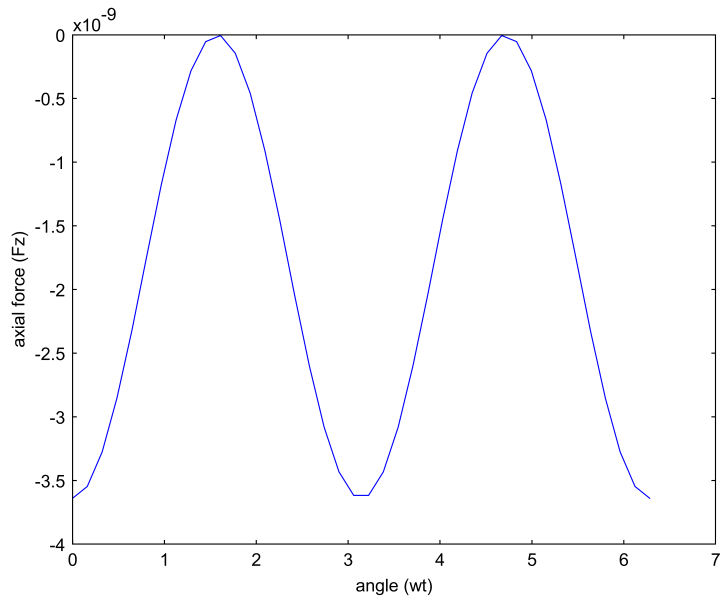


If you are running COMSOL Multiphysics with COMSOL Script or MATLAB, you can visualize the force over one period.

1 Select **Export>FEM structure** from the **File** menu.

2 Use the following commands to show the magnetic force on the plate over one period.

```
wt = linspace(0,2*pi,40);
for i = 1:length(wt)
  Fv(i) = postint(fem,'-2*pi*r*real(Jiphi_emqa)*real(Br_emqa)',...
    'dl',2,'phase',wt(i));
end
figure
plot(wt,Fv)
xlabel('angle (wt)'),ylabel('axial force (Fz)')
```



Transient Analysis

The previous model was an example of harmonic field variations. To study a transient case where the current increases abruptly from zero to a constant value, you only need to make a few alterations. To adapt the application mode to a transient analysis, you must first change the analysis type.

COMPUTING THE SOLUTION

1 Open the **Solver Parameters** dialog box and set the **Analysis** to **Transient**. Enter the vector `0 logspace(-4,-1,10)` in the **Times** edit field.

2 Set the **Absolute tolerance** to `1e-8`.

3 Solve the problem.

POSTPROCESSING AND VISUALIZATION

In a way similar to the harmonic analysis, you can visualize the magnetic flux density, the induced currents, and the magnetic forces.

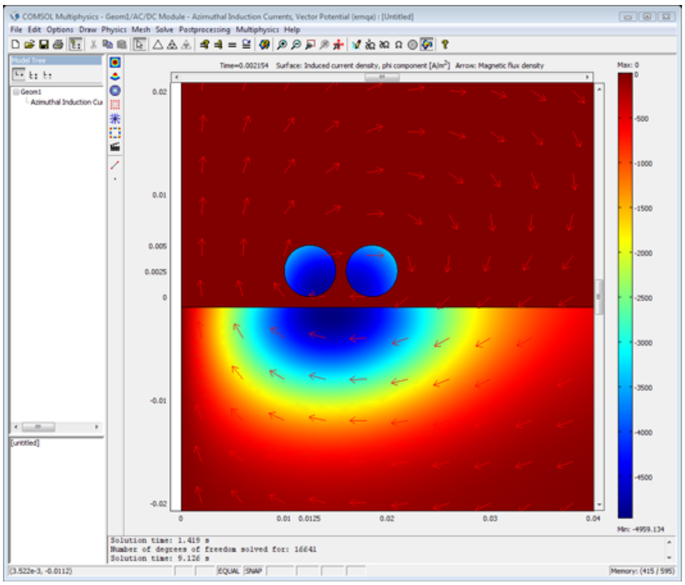
1 Open the **Plot Parameters** dialog box.

2 On the **General** tab, select **Surface** and **Arrow** plots.

3 On the **Arrow** tab, set the **Arrow data** to the magnetic flux density. Choose **Normalized** as **Arrow length**, set the **Arrow positioning** to 25 in both directions, and enter `0.5` as **Scale factor**.

4 Use the zoom features to see the eddy current distribution.

5 Change the evaluation time under the **General** tab in the **Plot Parameters** dialog box, or click **Start Animation** under the **Animate** tab, to see how the current distribution changes in time.

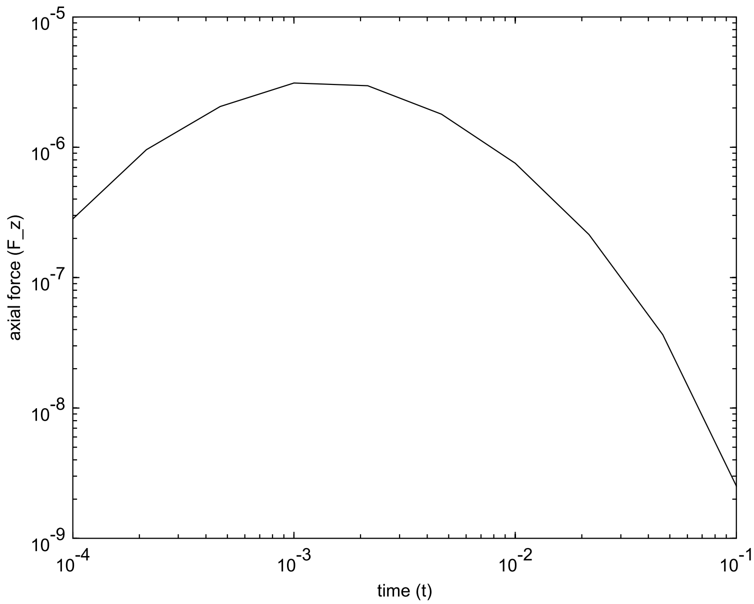


Modeling Using the Programming Language

```
% Start modeling by defining the names of the cylindrical
% coordinates and by defining the geometry of the structure.
clear fem
fem.sdim={'r' 'z'};
fem.geom=rect2(0,0.1,-0.05,0.05)+...
        rect2(0,0.08,-0.019,-0.001)+...
        circ2(0.0125,0.0025,0.0025)+...
        circ2(0.0185,0.0025,0.0025);

% Create the application structure.
clear appl
appl.mode.class='AzimuthalCurrents';
appl.mode.type='axi';
appl.prop.analysis='harmonic';
```

```
appl.var.nu=50;
appl.bnd.type={'ax' 'A0'};
appl.bnd.ind=[1 2 1 0 1 0 2 0 2 0 0 0 0 0 0 0 0];
appl.equ.sigma={'0' '5.99e7' '5.99e7'};
appl.equ.Vloop={'0' '0' '1e-4'};
appl.equ.ind=[1 2 3 3];
fem.appl=appl;
fem=multiphysics(fem);

% Create the mesh.
fem.mesh=meshinit(fem);
fem.mesh=meshrefine(fem);
fem.xmesh = meshextend(fem);

% Solve and visualize the solution.
fem.sol=femlin(fem);
postplot(fem,'tridata','Jiphi',...
        'arrowdata',{'Br' 'Bz'},'arrowscale',0.3,...
        'arrowstyle','normal')

% You can handle the postprocessing in exactly the same way as
% above, when solving the problem in the graphical user interface.
wt=linspace(0,2*pi,40);
for i=1:length(wt)
  Fv(i)=postint(fem,'-2*pi*r*real(Jiphi)*real(Br)','dl',2,...
    'phase',wt(i));
end
figure
plot(wt,Fv)
xlabel('angle (wt)'),ylabel('axial force (Fz)')

% To simulate the transient case, the analysis type has
% to be modified.
fem.appl.prop.analysis='transient';

% Generate the PDE coefficients for the transient analysis.
fem=multiphysics(fem);

% Regenerate the extended mesh.
fem.xmesh = meshextend(fem);

% To solve the problem, the solver femtime is invoked. You can
% visualize the result with the same command as in the time
% harmonic case. To study the result at a specific output time,
% set the parameter solnum.
fem.sol=femtime(fem,'tlist',[0 logspace(-4,-1,10)],'atol',1e-8);
postplot(fem,'tridata','Jiphi',...
        'arrowdata',{'Br' 'Bz'},'arrowscale',0.3,...
        'arrowstyle','normal','solnum',6)
```

```
% The axial force is visualized with the following commands.
F=postint(fem,'abs(2*pi*r*Jiphi*Br)',...
  'dl',2,'solnum',1:length(fem.sol.tlist));
loglog(fem.sol.tlist(:),F(:),'k');
xlabel('time (t)');
ylabel('axial force (F_z)');
```



Magnetic Field of a Helmholtz Coil

Introduction

A Helmholtz coil is a parallel pair of identical circular coils spaced one radius apart and wound so that the current flows through both coils in the same direction. This winding results in a uniform magnetic field between the coils with the primary component parallel to the axis of the two coils. The uniform field is the result of the sum of the two field components parallel to the axis of the coils and the difference between the components perpendicular to the same axis.

The purpose of the device is to allow scientists and engineers to perform experiments and tests that require a known ambient magnetic field. Helmholtz field generation can be static, time-varying DC, or AC, depending on the applications.

Applications include cancelling the earth's magnetic field for certain experiments; generating magnetic fields for determining magnetic shielding effectiveness or susceptibility of electronic equipment to magnetic fields; calibration of magnetometers and navigational equipment; and biomagnetic studies.

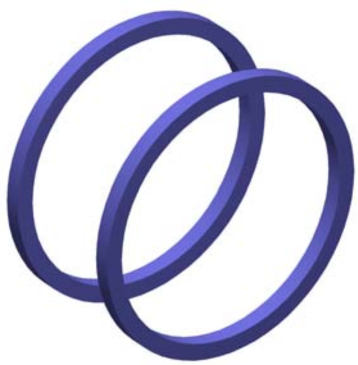


Figure 4-6: The Helmholtz coil consists of two coaxial circular coils, one radius apart along the axial direction. The coils carry parallel currents of equal magnitude.

Model Definition

The model is built using the 3D Magnetostatic application mode. The model geometry is shown in Figure 4-7.

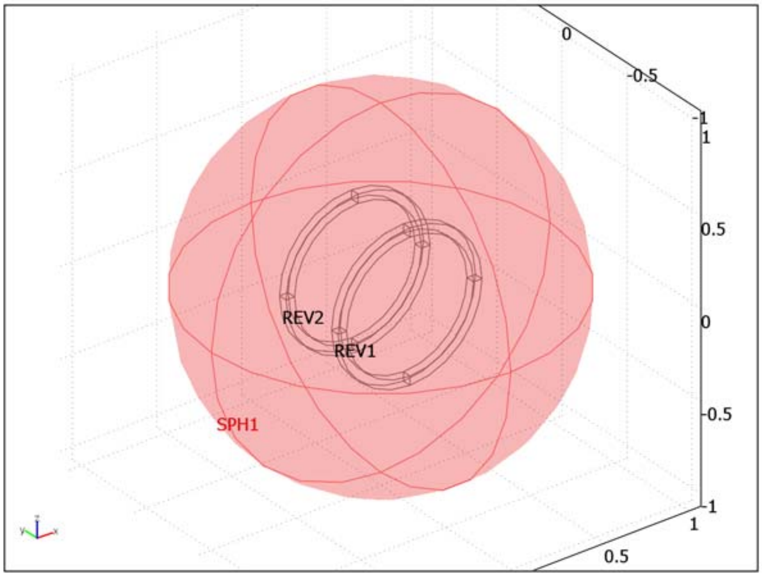


Figure 4-7: The model geometry.

DOMAIN EQUATIONS

Assuming static currents and fields, the magnetic vector potential **A** must satisfy the following equation:

$$\nabla \times (\mu^{-1} \nabla \times \mathbf{A}) = \mathbf{J}^e$$

where μ is the permeability, and $\mathbf{J}^e$ denotes the externally applied current density.

The relations between fields and potentials are given by

$$\mathbf{B} = \nabla \times \mathbf{A}$$

$$\mathbf{H} = \mu^{-1} \mathbf{B}$$

This model uses the following parameter value:

$$\mu = 4\pi \cdot 10^{-7}\ \mathrm{H/m}$$

The external current density is zero except in the circular coils, where a current density of 1 A/m^2 is specified. This corresponds to a coil current of 2.5 mA. The currents are specified to be parallel for the two coils.

To avoid numerical instability, the application mode by default use an extra equation that sets the divergence of the **A** field to zero (gauge fixing). This model uses a more efficient approach where the divergence of **A** is numerically adjusted to zero by using special pre- and postsmoothers called SOR gauge. See "Solver Settings for Numerical Gauge Fixing in Magnetostatics" on page 92 in the *AC/DC Module User's Guide* for more details on this topic.

BOUNDARY CONDITIONS

The only boundary conditions that you need to specify is for the exterior boundary, that is, the spherical surface, where you apply conditions corresponding to zero magnetic flux:

$$\mathbf{n} \times \mathbf{A} = 0$$

Results and Discussion

Figure 4-8 shows the magnetic flux density between the coils. You can see that the flux is fairly uniform between the coils, except for the region close to the edges of the coil.

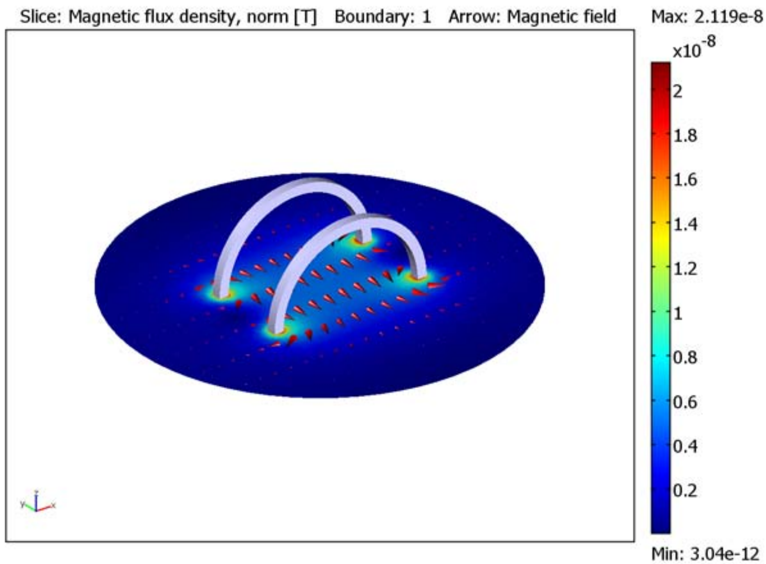


Figure 4-8: The surface color plot shows the magnetic flux density. The arrows indicate the magnetic field (H) strength and direction.

The main property of the Helmholtz coil is that the magnetic flux becomes uniform in a reasonably large region with a rather simple coil system. This is illustrated in Figure 4-9 (a), which shows a radial flux density profile for an axial position right between the coils. Figure 4-9 (b) shows an axial magnetic flux density profile. The model clearly demonstrates the highly uniform magnetic field obtained in a Helmholtz coil.

In the absence of any test object, this model is fully axisymmetric and could be implemented as a 2D axisymmetric model, which would be much less computationally demanding. However, this full 3D model has the advantage that a non axisymmetric test object could be included in the analysis as a slight modification of the model.

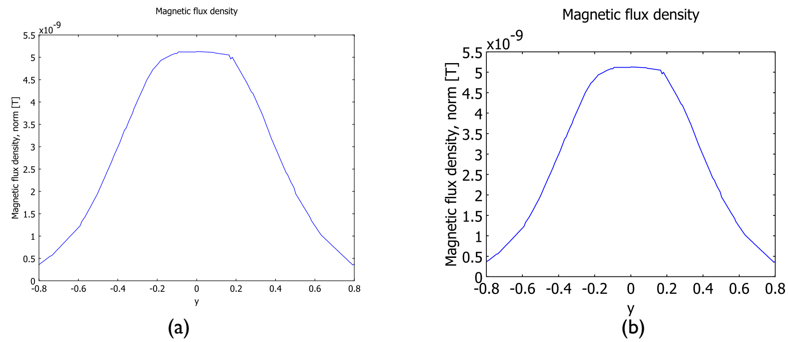


Figure 4-9: The magnetic flux density profile. In the graph at the left, the profile is taken along a radial cross sectional line through the axis right between the coils. In the graph on the right, the profile is taken along the axis. The high degree of uniformity is clearly shown.

Model Library path: `ACDC_Module/Electrical_Components/helmholtz_coil`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **3D** in the Space dimension list.

2 In the list of application modes, select **AC/DC Module>Statics>Magnetostatics**.

3 Click **OK**.

OPTIONS AND SETTINGS

From the **Options** menu, open the **Constants** dialog box and enter the following name, expression, and description (optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`J0`	`1[A/m^2]`	`Current density in coil`

GEOMETRY MODELING

The coils are made from squares in a 2D work plane, which you then revolve into a 3D geometry.

1 From the **Draw** menu open the **Work-Plane Settings** dialog box. Click **OK** to obtain the default work plane in the *xy*-plane.

2 In the 2D geometry, choose **Options>Axes/Grid Settings** and specify the settings according to the following table (you must clear the **Auto** check box on the **Grid** page to be able to enter the grid settings).

AXIS		GRID	
x min	`-0.6`	x spacing	`0.4`
x max	`-0.2`	Extra x	
y min	`-0.4`	y spacing	`0.2`
y max	`0.4`	Extra y	`-0.225 -0.175 0.175 0.225`

3 Draw a square with corners at (–0.425, 0.175) and (–0.375, 0.225).

4 Draw another square with corners at (–0.425, –0.225) and (–0.375, –0.175).

5 Select both squares and use the **Revolve** dialog box available from the **Draw** menu to revolve into the 3D geometry.

6 Finally add a sphere with radius 1 and center at the origin to the 3D geometry.

PHYSICS SETTINGS

Subdomain Settings

Apply the current in the coils as a given source current. Open the **Subdomain Settings** dialog box from the **Physics** menu and enter the source current according to the following table.

SETTING	SUBDOMAIN 1	SUBDOMAINS 2, 3
$\mathbf{J}^e$	`0 0 0`	`-J0*z/sqrt(x^2+z^2) 0 J0*x/sqrt(x^2+z^2)`

Boundary Conditions

Use the default **Magnetic insulation** boundary condition.

MESH GENERATION

1 Open the **Free Mesh Parameters** dialog box and select **Coarser** from the **Predefined mesh sizes** list on the **General** tab. Click the **Subdomain** tab, select the Subdomains 2 and 3, and set the **Maximum element size** to `0.05`.

2 Click **Remesh** to generate the mesh, then click **OK**.

COMPUTING THE SOLUTION

Because the default solver settings for this model efficiently handle the gauge fixing with the SOR gauge smoothers for the geometric multigrid preconditioner, simply click the **Solve** button on the Main toolbar to start solving.

POSTPROCESSING AND VISUALIZATION

1 Suppress the surface of the sphere, that is, Boundaries 1, 2, 3, 4, 21, 22, 31, and 32 using the **Suppress Boundaries** dialog box in the **Options** menu.

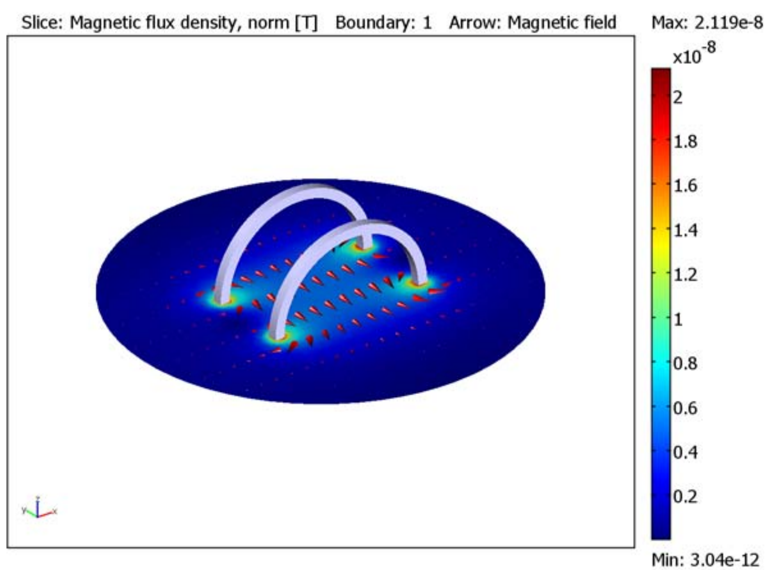
2 On the **General** page of the **Plot Parameters** dialog box select the **Slice**, **Boundary**, and **Arrow** check boxes and clear the **Geometry edges** check box.

3 On the **Slice** page use the default **Slice data**, which is **magnetic flux density, norm**. Set the number of **x levels** to `0` and the number of **z levels** to `1`.

4 On the **Boundary** page set **Boundary data** to `1` and select a **Uniform color** of your choice.

5 On the **Arrow** page select **Magnetic field** as **Arrow data**, set the **x points** to `24`, the **y points** to `10`, and the **z points** to `1`. Enter `0.5` as **Scale factor**.

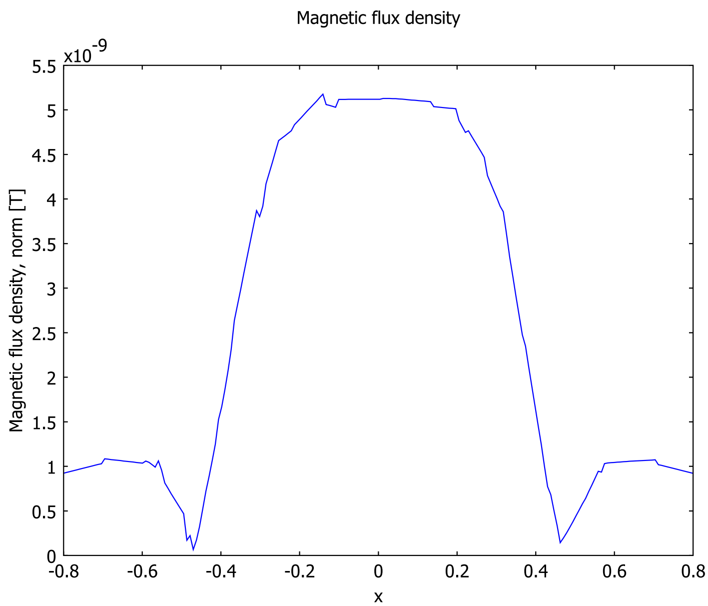
6 To add some lighting to the plot open the **Visualization/Selection Settings** dialog box from the **Options** menu and select **Scenelight** on the **Lighting** page. Disable lights 1 and 3.



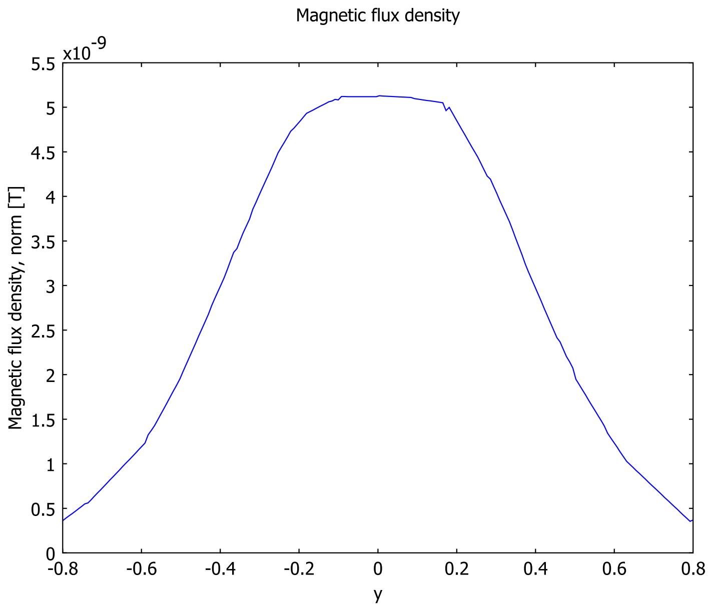
You can visualize the magnetic flux density profile using cross-section plots.

1 Open the **Cross-Section Plot Parameters** dialog box. On the **General** page click the **Title/Axis** button and set the title to `Magnetic flux density`.

2 On the **Line** tab use the default **y-axis data** which is **Magnetic flux density, norm**. Set the **x-axis data** to `x` and set **x0** to `-0.8` and **x1** to `0.8`. Click **Apply** to make a plot of the magnetic flux density along a line through the axis right between the coils.



To make a plot in the other direction change the **x-axis data** to `y` and set **x0** to `0`, **x1** to `0`, **y0** to `-0.8` and **y1** to `0.8`.



Inductance in a Coil

Introduction

This is a model of a simple AC coil: a single-turn thick copper wire. A parametric study shows the current distribution in the coil at different frequencies. The skin effect becomes increasingly important at the higher frequencies.

Model Definition

You build the model with the Axisymmetric Quasi-statics Azimuthal Currents application mode, using a time-harmonic formulation. The modeling takes place in the rz-plane. The modeling domain only includes the part where $r > 0$, in a manner such that revolving this view around the z-axis retrieves the full 3D geometry.

DOMAIN EQUATIONS

The dependent variable in this application mode is the azimuthal component of the magnetic vector potential **A,** which obeys the following relation:

$$(j\omega\sigma - \omega^2\varepsilon)A_\varphi + \nabla \times (\mu^{-1}\nabla \times A_\varphi) = \frac{\sigma V_{\mathrm{loop}}}{2\pi r}$$

where ω denotes the angular frequency, σ the conductivity, μ the permeability, ε the permittivity, and V_{loop} the voltage applied to the coil. Outside the coil, σ is set to zero.

BOUNDARY CONDITIONS

The model includes boundary conditions for the exterior boundary and the symmetry axis. For the exterior boundary, use magnetic insulation. This implies that the magnetic vector potential is zero at the boundary, corresponding to a zero magnetic flux. For the symmetry boundary use a symmetry condition. The magnetic potential is continuous across the interior boundary between the copper wire and the surrounding air.

Results and Discussion

Figure 4-10 and Figure 4-11 show the total current density in the coil at frequencies of 10 Hz and 2 kHz, respectively.

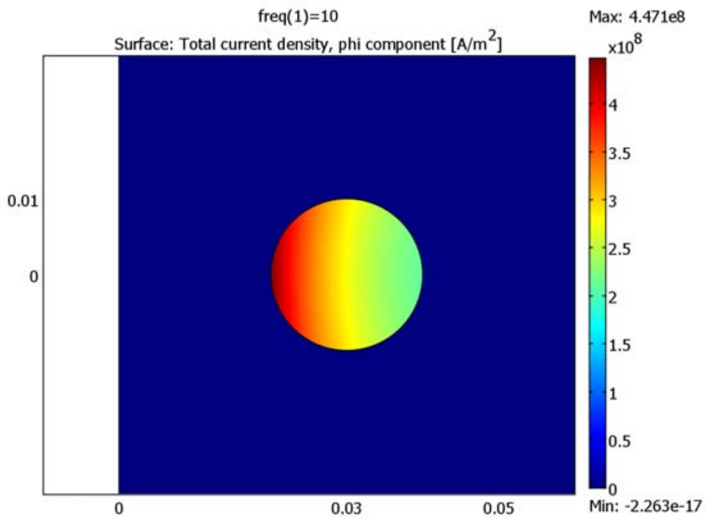


Figure 4-10: The current density at 10 Hz.

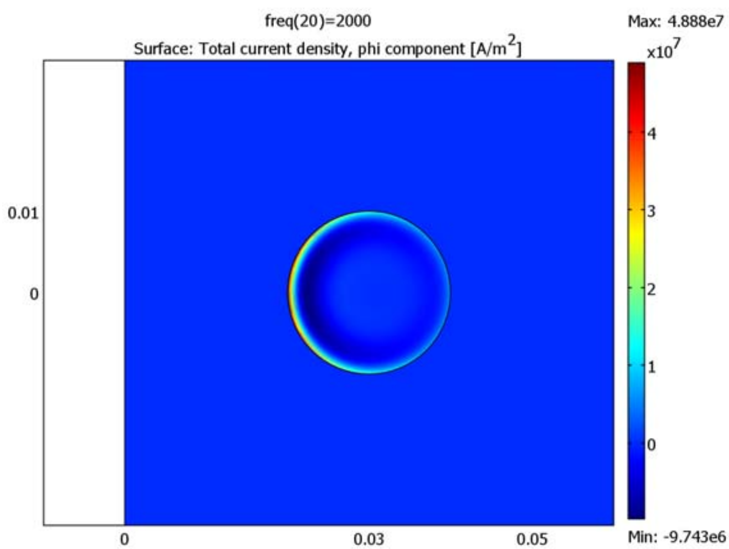


Figure 4-11: The current density at 2 kHz.

At 10 Hz, the current density is largely inversely proportional to the radial distance; at 2 kHz, the current is concentrated to the surface of the coil due to the skin effect.

The inductance versus frequency appears in Figure 4-12. The model discusses two alternative ways to calculate the inductance: the principle of virtual work and Ohm's law. The difference between the two approaches is less than 0.01 percent.

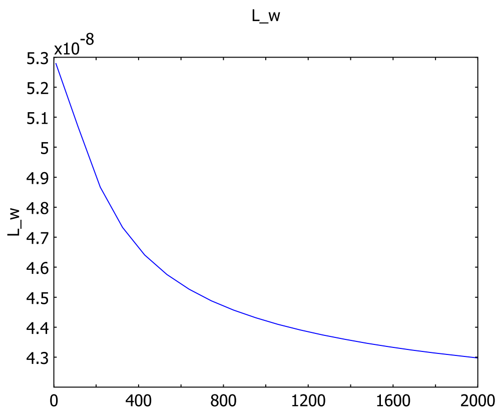


Figure 4-12: The inductance as a function of the frequency calculated using the virtual work method.

Model Library path: `ACDC_Module/Electrical_Components/inductance`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 On the **New** tab of the **Model Navigator**, select **Axial symmetry (2D)** in the **Space dimension** list.

2 Click on **AC/DC Module**, then select **Quasi-Statics, Magnetic> Azimuthal Induction Currents, Vector Potential>Time-harmonic analysis**.

3 Click **OK**.

OPTIONS AND SETTINGS

Start your modeling session by adjusting the main axes area to the size of the geometry to draw. You can also make your simulation easier by defining variables that you can use later when defining your problem.

1 Select **Axes/Grid Settings** from the **Options** menu to open the **Axes/Grid Settings** dialog box.
2 On the **Axis** tab set **r min** to `-0.25` and **r max** to `0.25`. Set the same interval limits for the z axis.
3 Now select the **Grid** tab and then clear the **Auto** check box, to manually define new grid settings.
4 Set **r spacing** to `0.05` and give the value `0.03` in the **Extra r** text field. Set **z spacing** to `0.05`, and **Extra z** to `0.01`.
5 Click **Apply** to see how the new settings are invoked. Note that the r axis settings are adjusted to keep the correct aspect ratio.
6 To define global constants, select **Constants** from the **Options** menu. This will open the **Constants** dialog box. This model uses only one constant parameter, namely the voltage applied to the coil.
7 Give **Name** and **Expression** according to the table below, by first entering the constant name and then the expression in the edit fields; the description is optional.

NAME	EXPRESSION	DESCRIPTION
`V0`	`1[V]`	`Voltage applied to coil`

8 The numerical value of the voltage that will be used in the model is now visible in the dialog box.
9 Click **OK** to close the dialog box.

GEOMETRY MODELING

You can now define the geometry of the structure, using the CAD tools that are available in COMSOL Multiphysics.

1 Start by drawing the half circle representing the air domain. A half circle is conveniently created as the union of a circle and a rectangle. First draw the circle. This is done by selecting **Draw Objects** and then **Ellipse/Circle (Centered)** from the **Draw** menu or by pressing the **Draw Centered Ellipse** button on the draw toolbar to the left of the main axes area. With the right mouse button, click and hold at (0, 0). Move the cursor to (0.2, 0) and release the button. The circle is now created.

2 Click the **Draw Rectangle** button on the draw toolbar or select the corresponding entry in the **Draw** menu. Draw a rectangle with opposite corners at (0, –0.2) and (0.2, 0.2), this time using the left mouse button.

3 Click the **Create Composite Object** button or select **Create Composite Object** in the **Draw** menu. In the dialog box that appears, mark C1 and R1 and click the **Intersection** button, or type `C1*R1` in the **Set formula** text field. Click **OK** to apply the changes and close the dialog box.

4 Continue by drawing the circle representing the coil. Select **Draw Centered Ellipse** again. With the right mouse button, click at (0.03, 0) and move the cursor to (0.03,0.01), where you release the button.

5 Double-click **GRID** in the status bar at the bottom of the GUI to hide the grid lines.

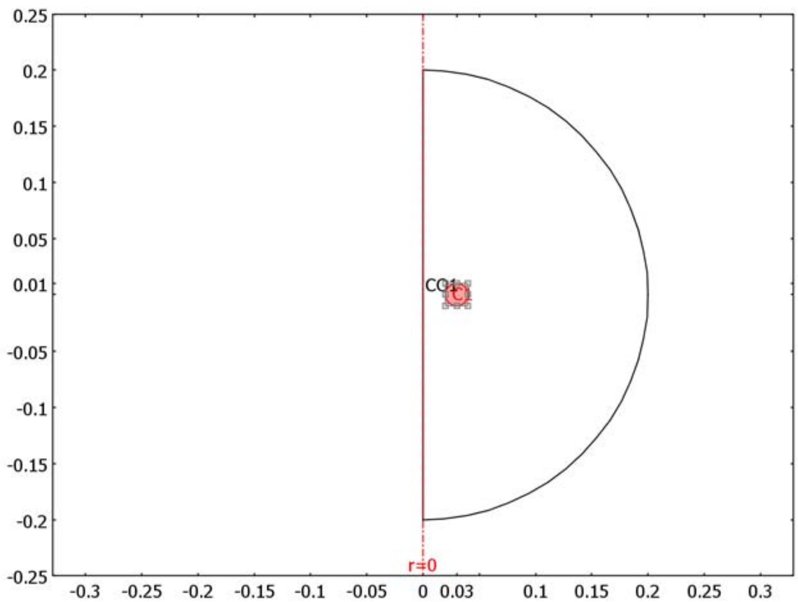


Figure 4-13: Geometry.

PHYSICS SETTINGS

Scalar Variables

1 Open the **Application Scalar Variables** dialog box by selecting **Scalar Variables** from the **Physics** menu.

2 The solution should be calculated for a range of frequencies. Type `freq` in the **Expression** field for `nu_emqa`. Leave the permittivity and the permeability of vacuum at their default values.

3 Click **OK**.

Subdomain Settings

1 Select **Subdomain Settings** from the **Physics** menu.

2 The domain properties on the **Electric Parameters** tab for this model are given in the following table. Use the default values for the properties that are not given here for Subdomain 2, and for all properties for Subdomain 1.

SETTINGS	SUBDOMAIN 2
σ	`5.998e7`
V_{loop}	`V0`

3 Click **OK**.

Boundary Conditions

1 Open the **Boundary Settings** dialog box by selecting **Boundary Settings** from the **Physics** menu

2 Give the boundary conditions according to the following table.

SETTINGS	BOUNDARY 1	BOUNDARIES 2, 3
Boundary condition	Axial symmetry	Magnetic insulation

3 Click **OK**.

Boundary 1 is the vertical boundary along the z-axis. The axial symmetry boundary condition makes sure that the solution is symmetric around this axis. The magnetic insulation condition sets the magnetic potential A_φ to zero along the Boundaries 2 and 3.

Coupling Variables

Scalar coupling variables are used for integrating the current density and the magnetic energy, in order to calculate the inductance and resistance of the coil.

1 Select **Integration Coupling Variables>Subdomain Variables** from the **Options** menu to open the **Subdomain Integration Variables** dialog box.

2 Specify the variable names and expressions according to the following tables. Use the default **Integration order** 4 and keep the **Destination global** check boxes selected. Put each coupling variable on a separate row in the table on the **Source** tab.

SETTING	SUBDOMAINS 1, 2
`Wm`	`2*pi*r*Wmav_emqa`

SETTING	SUBDOMAIN 2
`Itot`	`Jphi_emqa`

3 Click **OK**.

Expression Variables

Expression variables are used for completing the inductance and resistance calculations.

1 Select **Expressions>Scalar Expressions** from the **Options** menu to open the **Scalar Expressions** dialog box.

2 Specify the variable names and expressions according to the following table.

NAME	EXPRESSION
`L_w`	`4*Wm/abs(Itot)^2`
`L_e`	`imag(V0/Itot)/(2*pi*freq)`
`Resist`	`real(V0/Itot)`

3 Click **OK**.

MESH GENERATION

In this model, as in many others dealing with electromagnetic phenomena, the effects on the fields near interfaces between materials are of special interest. To get accurate results make sure you generate a mesh that is very fine in these areas. To achieve this, refine the mesh once.

1 Select **Initialize Mesh** in the **Mesh** menu or use the corresponding button on the main toolbar above the main axes area to generate a mesh.

2 Select **Refine Mesh** in the **Mesh** menu or use the corresponding button on the main toolbar.

3 To see the mesh in the interesting region better, select **Zoom Window** from the **Options** menu. You can now draw a rectangular window around the coil and the cylinder to get a better view of the mesh.

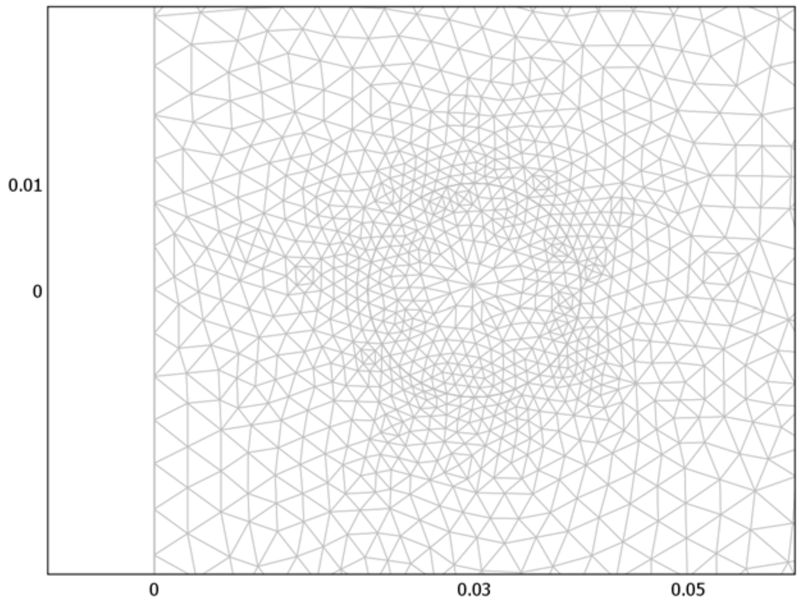


Figure 4-14: The mesh for the inductance computation.

COMPUTING THE SOLUTION

1 Open the **Solver Parameters** dialog box by selecting **Solver Parameters** in the **Solve** menu, or by clicking the **Solver Parameters** button. In the **Solver** list, select **Parametric**. Type `freq` in the **Parameter name** edit field, and `linspace(10,2000,20)` in the **Parameter values** edit field. This means that the solution is computed for 20 equally-spaced values of `freq`, running from 10 Hz through 2000 Hz.

2 Click **OK** to close the dialog box.

3 Select **Solve** in the **Solve** menu, or click the **Solve** button, to compute the solution.

POSTPROCESSING AND VISUALIZATION

After the solution is calculated, a surface plot is automatically shown for the magnetic flux density. The buttons on the plot toolbar are shortcuts to get plots of other types. Full control over the plot generation is provided in the **Plot Parameters** dialog box.

1 Select **Plot Parameters** from the **Postprocessing** menu to open the **Plot Parameters** dialog box.

2 In the **Plot type** area on the **General** page, select the **Surface** and **Geometry edges** check boxes.

3 On the **Surface** page, set the **Surface Expression** from the list of predefined quantities to **Total current density, phi component (Jphi_emqa)**.

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

5 By default, the plot is rendered for the last entry in the frequency list. To view another solution, select the desired frequency from the **Parameter value** list on the **General** page. Click **OK** to generate the plot and close the dialog box.

The resistance and the inductance are calculated for all frequencies and can be visualized using point plots.

1 Open the **Cross-Section Plot Parameters** dialog box from the **Postprocessing** menu. On the **General** page, click **Point plot** and mark all the solutions.

2 On the **Point** page, type `Resist` in the **Expression** edit field. The coordinates can be left at their default values. The value of the resistance is evaluated at the specified coordinates, but `Resist` is a scalar variable and its value is therefore independent of which coordinates you use, as long as they are inside the geometry.

3 Click **Apply** to generate a plot of the resistance as a function of the frequency.

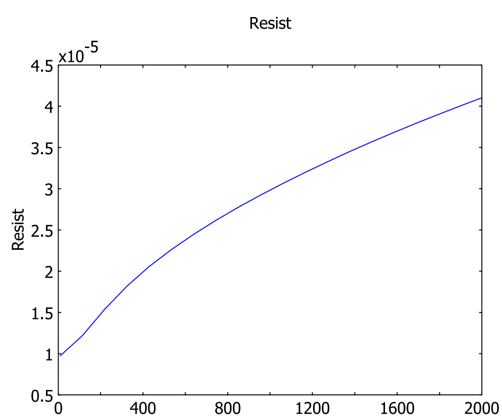


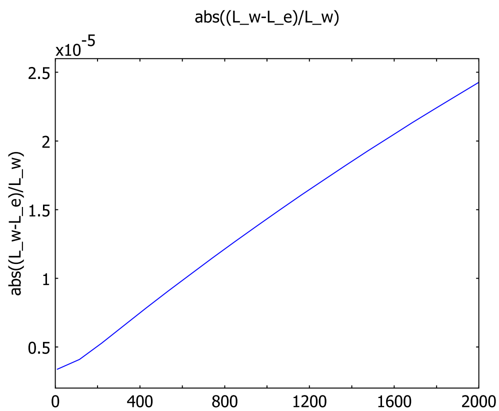
Figure 4-15: Resistance (Ω) as a function of frequency (Hz).

You can visualize the calculated values of the inductance in the same way as the resistance; Figure 4-12 on page 133, for example, shows the result for L_w, the inductance calculated using the method of virtual work. To reproduce this plot in a new figure window do as follows:

4. Click the **General** tab, then select **New figure** from the **Plot in** list.
5. Return to the **Point** page. In the **Expression** edit field type `L_w`, then click **Apply**.

   The relative difference between the results from the two calculation methods for the inductance gives an indication of the errors introduced from the discretization and the finite geometry. Visualize this entity by executing the following instructions:

6. Click the **General** tab, then select **New figure** from the **Plot in** list.
7. Return to the **Point** page. In the **Expression** edit field type `abs((L_w-L_e)/L_w)`, then click **OK**.



Integrated Square-Shaped Spiral Inductor

Introduction

This example presents a model of a micro-scale square inductor, used for LC bandpass filters in microelectromechanical systems (MEMS).

The modeling is carried out in two steps. Because this is a static simulation, it is possible to compute the electric and magnetic fields separately. The first step computes the currents in the inductor. You then introduce these currents as sources for the magnetic flux computations in a second step.

The purpose of the model is to calculate the self-inductance of the micro-inductor. Given the magnetic field, you can compute the self-inductance, L, from the relation

$$L = \frac{2W_m}{I^2}$$

where W_m is the magnetic energy, and I is the current. This model includes a boundary condition that sets the current to 1. The self-inductance L is the L_{11} component of the inductance matrix and is available as a predefined variable (`L11_emqa`) for postprocessing.

Model Definition

The model geometry consists of the spiral-shaped inductor and the air surrounding it. The subdomain corresponding to air can be done arbitrary large depending on the position of the device. Figure 4-16 below shows the inductor and air subdomains used in the model. The outer dimensions of the model geometry are around 0.3 mm.

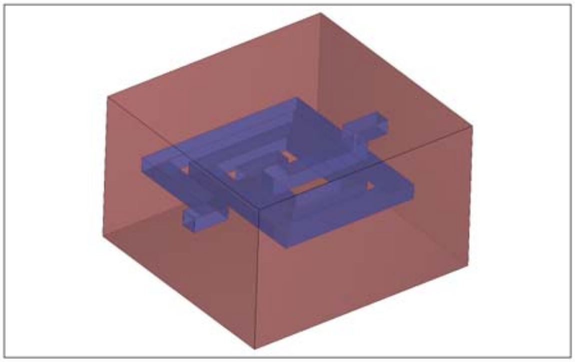


Figure 4-16: Inductor geometry and the surrounding subdomain.

The model equations are the following:

$$-\nabla \cdot (\sigma \nabla V - \mathbf{J}^e) = 0$$
$$\nabla \times (\mu_0^{-1} \mu_r^{-1} \nabla \times \mathbf{A}) + \sigma \nabla V = \mathbf{J}^e$$

In the equations above, σ denotes the electric conductivity, $\mathbf{A}$ the magnetic vector potential, V the electric scalar potential, $\mathbf{J}^e$ the externally generated current density vector, μ_0 the permittivity in vacuum, and μ_r the relative permeability.

The electric conductivity in the coil is set to 10^6 S/m and 1 S/m in air. The conductivity of air is arbitrarily set to a small value in order to avoid singularities in the solution, but the error becomes small as long as the value of the conductivity is small.

The constitutive relation is specified with the following expression:

$$\mathbf{B} = \mu_0 \mu_r \mathbf{H}$$

where $\mathbf{H}$ denotes the magnetic field.

The boundary conditions are of three different types corresponding to the three different boundary groups; see Figure 4-17 (a), (b), and (c) below.

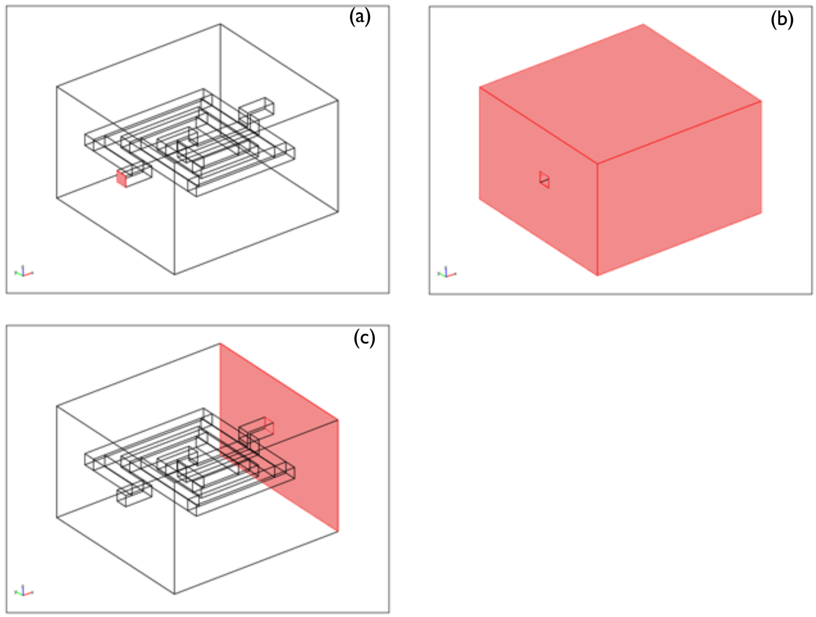


Figure 4-17: Boundaries with the same type of boundary conditions.

The boundary condition for the boundary highlighted in Figure 4-17 (a) is a magnetic insulating boundary with a port boundary condition. For the boundaries in Figure 4-17 (b), both magnetic and electric insulation prevail. The last set of boundary conditions, Figure 4-17 (c), are magnetically insulating but set to a constant potential of 0 V (ground).

Results

Figure 4-18 shows the electric potential in the inductor and the magnetic flux lines. The thickness of the flow lines represents the magnitude of the magnetic flux. As expected this flux is largest in the middle of the inductor.

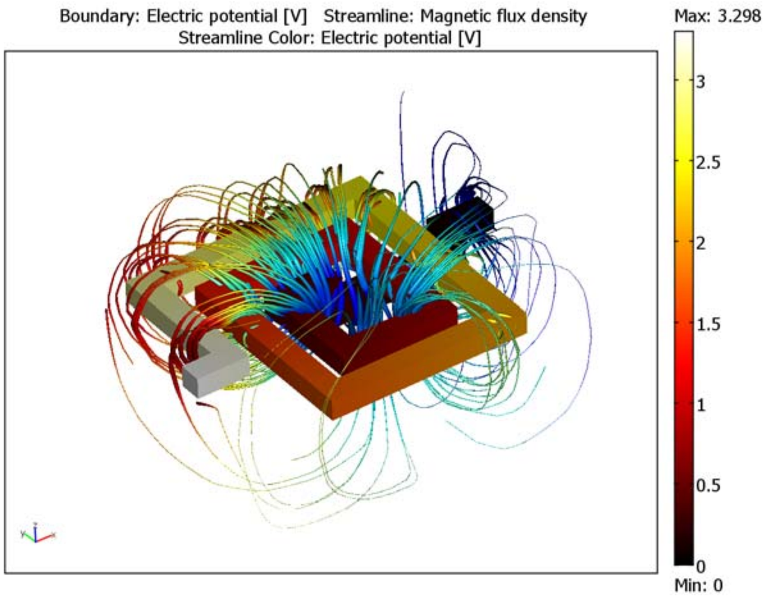


Figure 4-18: Electric potential in the device and magnetic flux lines around the device.

The model gives a self-inductance of 7.21e-10 H.

Model Library path: `ACDC_Module/Electrical_Components/inductor`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Start COMSOL Multiphysics.

2 In the **Model Navigator** click the **Multiphysics** button.

3 Select **3D** from the **Space dimension** list.

4 Select the **AC/DC Module>Statics>Magnetostatics** application mode.

5 Click the **Add** button.

6 Open the **Application Mode Properties** dialog box by clicking the corresponding button.

7 Set the property **Potentials** to **Electric and magnetic**.

8 Click **OK**.

OPTIONS AND SETTINGS

Give the axis settings according to the following table.

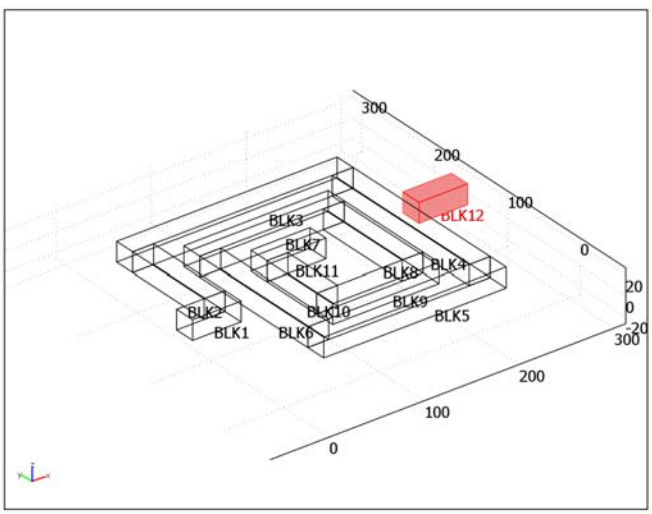
AXIS	MIN	MAX
x	-50	300
y	-50	300
z	-25	25

GEOMETRY MODELING

1 Start the modeling of this geometry by creating a number of solid block objects that form the base of the inductor. Use the data for the blocks according to the table below.

NAME	LENGTH			AXIS BASE POINT		
	X	Y	Z	X	Y	Z
BLK1	51	22	20	-20	99	0
BLK2	22	98	20	9	121	0
BLK3	232	22	20	9	219	0
BLK4	22	188	20	219	31	0
BKL5	192	22	20	49	9	0
BLK6	22	148	20	49	31	0
BLK7	152	22	20	49	179	0
BLK8	22	108	20	179	71	0
BLK9	112	22	20	89	49	0
BLK10	22	68	20	89	71	0
BLK11	62	22	20	89	139	0
BLK12	51	22	20	249	139	0

The geometry in the drawing area should now look like that in the following picture.

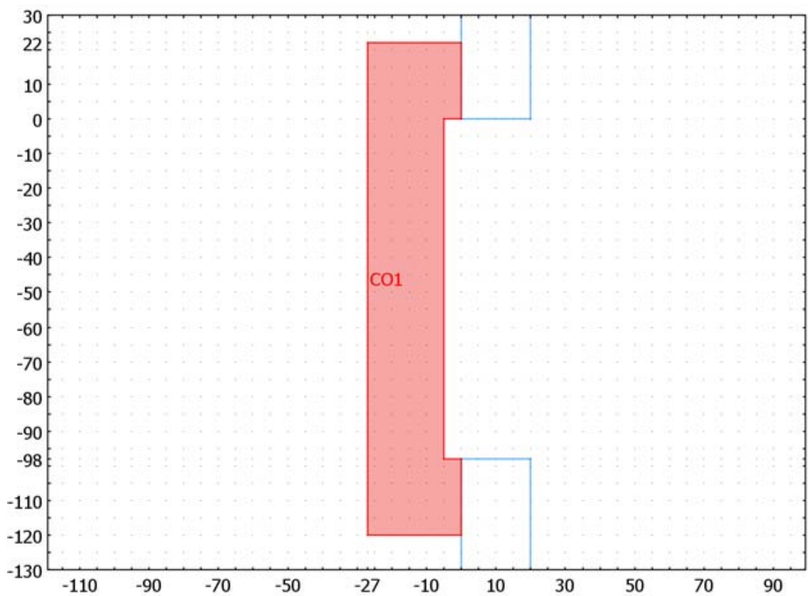


2 From the **Draw** menu, open the **Work-Plane Settings** dialog box and choose **BLK12** face number 3 on the **Face Parallel** tab.

3 Give the axis and grid setting as in the following table.

AXIS		GRID	
x min	-30	x spacing	5
x max	10	Extra x	-27
y min	-130	y spacing	5
y max	30	Extra y	-98 22

4 Click the **Line** button on the Draw toolbar, and click at (–27, –120), (–27, 22), (0, 22), (0, 0), (–5, 0), (–5, –98), (0, –98), and (0, –120). Finally right-click to create a composite solid object.



5 Open the **Extrude** dialog box and set the **Distance** to 22. Click **OK** to create the extruded object.

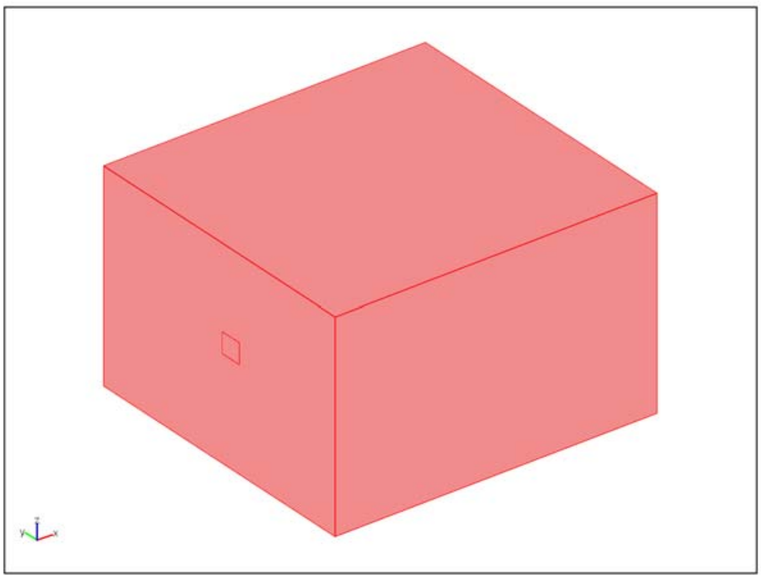
6 Select all geometry objects, by using the **Select all** option from the **Edit** menu. Then open the **Create Composite Object** dialog box. Be sure to clear the **Keep interior boundaries** check box, and verify that all objects are selected. Then click **OK** to create the composite object as the union of all objects.

7 Now, draw a block that surrounds the inductor. Set the length of the sides to 320, 300, and 200, for the x-, y-, and z-coordinates respectively. Set the axis base point to (–20, –25, –100).

8 Select both the block object and the composite object CO1 by using the **Shift** key.

9 Open the **Create Composite Object** dialog box. Select the **Keep interior boundaries** check box and click **OK**.

10 All distances in this model are given in µm, so you must scale the geometry with a factor of 10^{-6} in order to obtain the correct size. Do this by clicking the **Scale** button, and then giving the value `1e-6` for all three scaling factors. After that, click the **Zoom Extents** button on the Main toolbar to be able to see the object.



PHYSICS SETTINGS

Subdomain Settings

Enter the following values in the **Subdomain Settings** dialog box and then click **OK**.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2
σ	1	1e6

Setting the conductivity to zero in air (Subdomain 1) would lead to a numerically singular problem. Therefore the conductivity is arbitrarily set to 1 S/m. Although not physically correct, this value is much smaller than the electric conductivity in the inductor, 10^6 S/m. The electric and magnetic fields are only marginally affected by the value of the conductivity in air as long as it is small.

Boundary Conditions

Choose **Boundary Settings** from the **Physics** menu, and set the boundary conditions according to the following table.

SETTINGS	BOUNDARY 5	BOUNDARIES 75,76	BOUNDARIES 1, 2, 3, 4, 10
Electric Parameters	Port	Ground	Electric insulation
Magnetic Parameters	Magnetic insulation	Magnetic insulation	Magnetic insulation
Port tab	Use port as inport		

The Boundaries 5 and 76 are the two ends of the inductor. When finished, click **OK**.

MESH GENERATION

1 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.
2 On the **Global** page, select **Coarser** from the **Predefined mesh sizes** list.
3 Click the **Subdomain** tab. Select Subdomain 2 and type `3e-5` in the **Maximum element size** edit field.
4 Click the **Remesh** button to generate the mesh. Click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to start solving

This model uses an efficient approach for handling the gauge fixing with the SOR gauge smoothers for the geometric multigrid preconditioner; see “Solver Settings for Numerical Gauge Fixing in Magnetostatics” on page 92 in the *AC/DC Module User’s Guide* for more information.

POSTPROCESSING AND VISUALIZATION

Now add a visualization of the magnetic flux density lines to illustrate the results from the simulation.

1 From the **Options** menu, choose **Suppress>Suppress Boundaries**.
2 In the **Suppress Boundaries** dialog box, select Boundaries 1–4, 10, and 75. Click **OK**.
3 Open the **Plot Parameters** dialog box and clear the **Slice** and **Geometry edges** check boxes, and select **Streamline** and **Boundary** as plot types.

4 On the **Boundary** page, set the **Boundary expression** to V.

5 Click the **Streamline** tab and choose the **Magnetic flux density** as **Streamline data** and set the **Number of start points** to 31.

6 Then change the **Line** type to **Tube** and click the **Tube Radius** button. Select the **Radius data** check box and select **Magnetic flux density, norm** as **Radius data**. Set the **Radius scale factor** to 0.7. Click **OK** to close the **Tube Radius Parameters** dialog box.

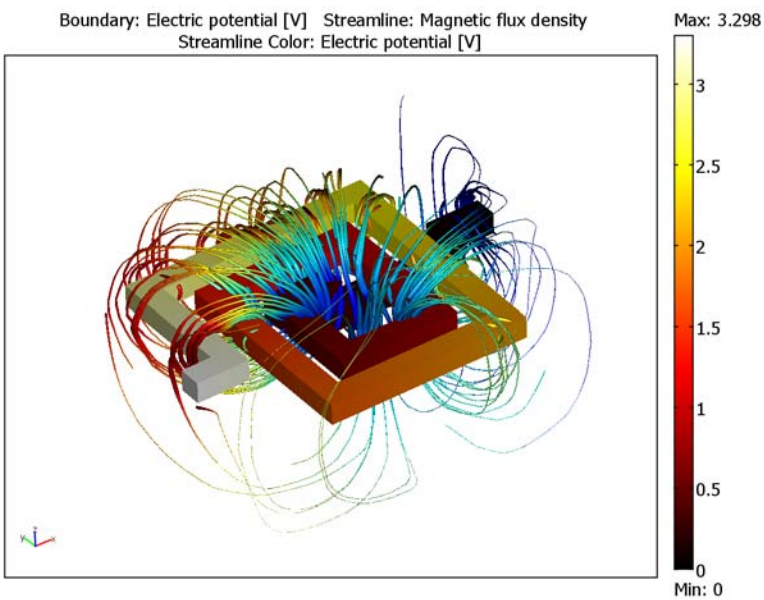
7 Click the **Use expression to color streamlines** option button.

8 Click the **Color Expression** button and select **Electric potential** as **Streamline color data**. Clear the **Color scale** check box. Click **OK** to close the **Streamline Color Expression** dialog box.

9 Click the **Advanced** button and set the **Maximum number of integration steps** to 100. Click **OK** to close the **Advanced Streamline Parameters** dialog box.

10 Click **OK** to close the **Plot Parameters** dialog box.

11 For improved visualization click the **Headlight** button on the Camera toolbar.



To extract the self-inductance, select **Data Display>Global** from the **Postprocessing** menu and then type L11_emqa in the **Expression** edit field, or:

1 Open the **Domain Plot Parameters** dialog box from the **Postprocessing** menu.

2 Click the **Point** tab and plot **Inductance matrix, element 11**.

Modeling Using the Programming Language

You can create the Inductor model from COMSOL Script or MATLAB using COMSOL Multiphysics command-line functions. The following script contains all the steps for defining the model, computing the solution, visualizing the results, and evaluating the inductance:

```
% Start by defining the geometry.
clear fem
g = { ...
    block3(51,22,20,'corner',[-20 99 0]),...
    block3(22,98,20,'corner',[9 121 0]),...
    block3(232,22,20,'corner',[9 219 0]),...
    block3(22,188,20,'corner',[219 31 0]),...
    block3(192,22,20,'corner',[49 9 0]),...
    block3(22,148,20,'corner',[49 31 0]),...
    block3(152,22,20,'corner',[49 179 0]),...
    block3(22,108,20,'corner',[179 71 0]),...
    block3(112,22,20,'corner',[89 49 0]),...
    block3(22,68,20,'corner',[89 71 0]),...
    block3(62,22,20,'corner',[89 139 0]),...
    block3(51,22,20,'corner',[249 139 0]),...
  };

l2 = line2([-25 -25 0 0 -5 -5 0 0],[-120 22 22 0 0 -98 -98 -120]);

g{end+1} = extrude(l2,'Distance',22,...
    'Wrkpln',[249 249 600;139 139 139;0 351 0]);

co1 = geomcomp(g,'face','all','edge','all');

b1 = block3(320,300,200,'corner',[-20 -25 -100]);

co2 = geomcomp({co1 b1});

fem.geom = scale(co2,1e-6,1e-6,1e-6);

% The physical properties of the model is defined in the
% application mode structure. Continue the modeling by creating
% this structure. The commands to use are given below.
clear appl

appl.mode='QuasiStatics';
appl.prop.analysis='static';

appl.equ.sigma={'1','1e6'};

appl.bnd.inport={'1','0','0'};
appl.bnd.eltype={'port','V0','nJ0'};
```

```
appl.bnd.magtype='A0';
appl.bnd.ind=zeros(1,76);
appl.bnd.ind([5])=1;
appl.bnd.ind([75 76])=2;
appl.bnd.ind([1 2 3 4 10])=3;

fem.appl=appl;

fem=multiphysics(fem);

% Generate the mesh, by issuing the following command.
fem.mesh=meshinit(fem, ...
  'hauto',7, ...
  'hmaxsub',[2,3e-5]);

% Generate the extended mesh structure.
fem.xmesh=meshextend(fem);

% You can now solve the problem.
fem.sol=femlin(fem);

% Visualize the result of the simulation, with the following
% properties to the plot function.
nBdl = setdiff(1:76,[1:4 10 75]);
postplot(fem,...
  'bdl',nBdl,...
  'cont', 'internal',...
  'tridata','V',...
  'tribar', 'on',...
  'flowtubescale',0.3,...
  'flowcolor','k',...
  'flowdata',{'Bx','By','Bz'},...
  'flowlines',50,...
  'flowback','off',...
  'flowsteps',150,...
  'scenelight','on',...
  'axisequal','on',...
  'axisvisible','off');

% The inductance can be calculated with the command
inductance=postinterp(fem,'L11',[0;0;0]);
```

Inductance of a Power Inductor

The model shows an inductance calculation on a large 3D geometry using higher-order vector elements and memory-efficient iterative solver settings.

Introduction

Power inductors are a central part of many low-frequency power applications. They are, for example, used in switched power supplies and DC-DC converters. The inductor is used in conjunction with a high-power semiconductor switch that operates at a certain frequency, stepping up or down the voltage on the output. The relative low voltage and high power consumption puts high demands on the design of the power supply and especially on the inductor, which must be designed with respect to switching frequency, current rating, and warm environments.

A power inductor usually has a magnetic core to increase its inductance value, reducing the demands for a high frequency while keeping the sizes small. The magnetic core also reduces the electromagnetic interference with other devices. There are only crude analytical formulas or empirical formulas available for calculating impedances, so computer simulations or measurements are necessary in the design of these inductors. This model uses a design drawn in an external CAD software, imports the geometry to COMSOL Multiphysics, and finally calculates the inductance from the specified material parameters and frequency.

Model Definition

The model uses the Quasi-static application mode taking electric induced and inductively induced currents into account. This formulation, often referred to as an AV formulation, solves both for the magnetic vector potential $\mathbf{A}$ and the electric potential V. In addition, it must also solve for the Gauge fixing on the vector potential,

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

At low frequencies the inductance is almost constant. For high frequencies the capacitive effects play a role, and the permeability usually decreases, causing a frequency-dependent inductance. A model limiting factor in increasing the frequency is the skin depth, so at a frequency in the vicinity of 10 kHz the model either needs a finer mesh at the conductor boundary or an impedance boundary condition. The following table lists the material properties used in this model.

MATERIAL PARAMETER	COPPER	CORE
σ	$5.997 \cdot 10^7$	10
ε_r	1	1
μ_r	1	10^3

Using a low conductivity for the surrounding air improves the stability of the iterative solver. This has negligible impact on the solution.

The outer boundaries are mainly magnetic insulation and electric insulation,

$$\mathbf{n} \times \mathbf{A} = \mathbf{0}$$
$$\mathbf{n} \cdot \mathbf{J} = 0$$

For the boundaries to the conductor, one end is grounded, and the other end has a port boundary condition. The port boundary condition gives the impedance of the inductor, and you can calculate the inductance from the formula,

$$L_{11} = \frac{\mathrm{Im}(Z_{11})}{\omega}$$

where ω is the angular frequency, and $\mathrm{Im}(Z_{11})$ is the imaginary part of the impedance.

COMPUTING THE SOLUTION

An analysis using the AV formulation with gauge fixing turned on consumes large amounts of memory when you use direct solvers like UMFPACK or SPOOLES. This model has almost 200,000 degrees of freedom, so an iterative solver is necessary. The geometric multigrid (GMG) solver with the Vanka pre- and postsmoother solves this problem efficiently. Different element order defines the multigrid hierarchy, which means that a direct solver solves the problem on linear vector and Lagrange elements. The iterative solver then produce the solution for the quadratic versions of these elements.

Results and Discussion

At a frequency of 1 kHz the inductance is 97 μH, and the figure below shows the electric potential and the magnetic flux density in a combined plot.

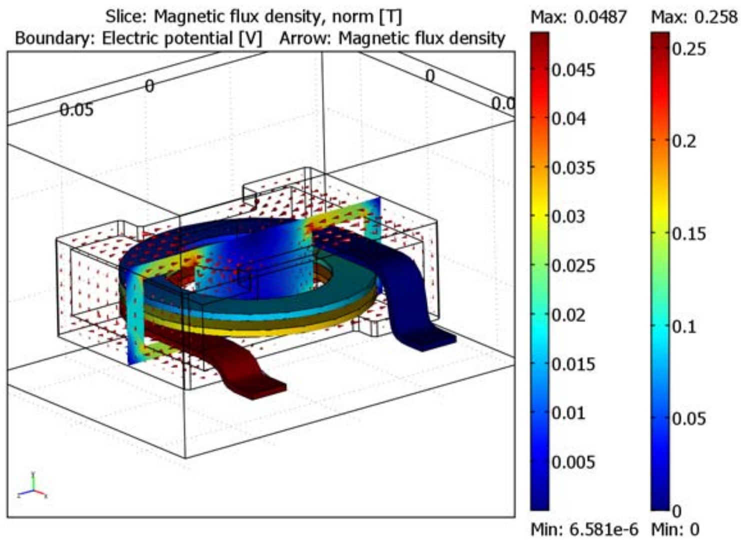


Figure 4-19: The final plot of the power inductor, showing the potential on the coil, the magnitude of the flux density inside the ferrite core, and the direction of the same as arrows.

Model Library path: `ACDC_Module/Electrical_Components/power_inductor`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **3D** in the **Space dimension** list.

2 In the list of application modes, select **AC/DC Module>Quasi-Statics, Electromagnetic>Electric and Induction Currents**.

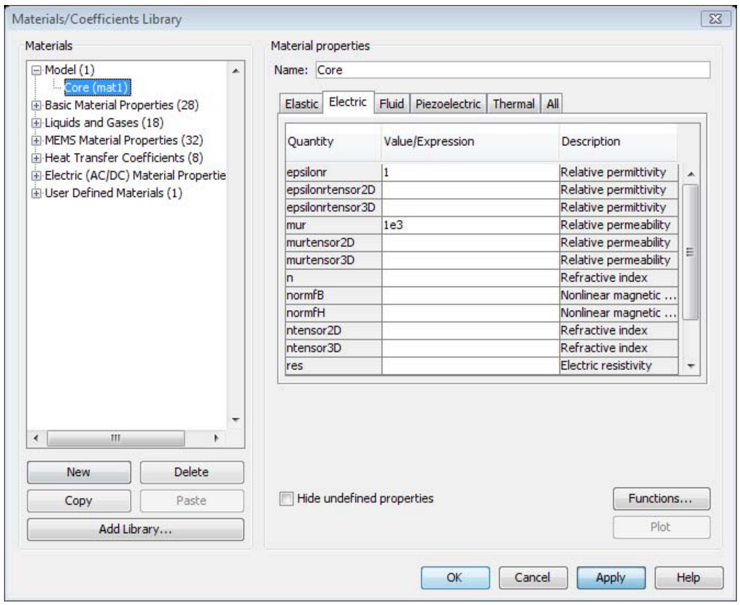
3 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Material/Coefficient Library**. In the dialog box that appears, click the **New** button.

2 Type `Core` in the **Name** edit field for the new material.

3 Click the **Electric** tab and type the values for each material parameter listed in the table below.



MATERIAL PARAMETER	CORE
σ	10
ε_r	1
μ_r	1e3

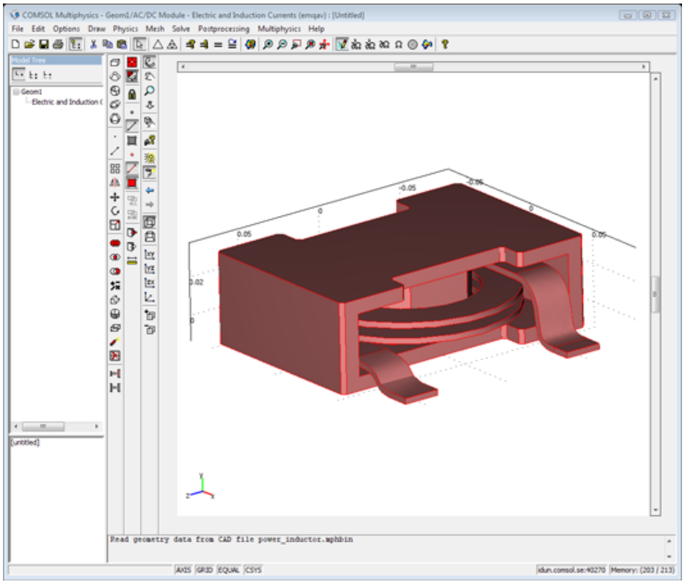
4 Click **OK**.

GEOMETRY MODELING

1 From the **File** menu, choose **Import>CAD Data From File**.

2 In the **Import CAD Data From File** dialog box, make sure that the **COMSOL Multiphysics file** or **All 3D CAD files** is selected in the **Files of type** list.

3 Locate the `power_inductor.mphbin` file in the model path specified on page 155, and click **Import**.



4 From the **Draw** menu, choose **Block**.

5 In the dialog box that appears, define the block properties according to the table below.

LENGTH X	LENGTH Y	LENGTH Z	BASE	AXIS BASE POINT (X, Y, Z)
`0.15`	`0.12`	`0.2`	`Corner`	`(-0.07,-0.031,-0.1)`

PHYSICS SETTINGS

Scalar Variables

1 Open the **Scalar Variables** dialog box from the **Physics** menu.

2 Type `1e3` for the frequency, **nu_emqav**. Click **OK**.

Subdomain Settings

1 Open the **Subdomain Settings** dialog box from the **Physics** menu.

2 Select Subdomain 1 and click **Electric Parameters** tab. Type `1` in the edit field for the electric conductivity.

3 Select Subdomain 2 and select **Core** from the **Materials** list. This is the material you defined in the Materials library earlier.

4 Select Subdomain 3 and click the **Load** button. Locate and select **Copper** in the dialog box and click **OK**.

5 Click **OK** again to close the **Subdomain Settings** dialog box.

Boundary Conditions

1 Open the **Boundary Settings** dialog box from the **Physics** menu.

2 Select all boundaries and make sure that the default **Magnetic insulation** is selected from the **Boundary condition** list.

3 Click the **Electric Parameters** tab. Select **Electric insulation** from the **Boundary condition** list.

4 Select Boundary 78, which is one of the boundaries for the inductor coil. Choose the **Ground** boundary condition from the **Boundary condition** list.

5 Select the other coil boundary, which is number 79. Select the **Port** boundary condition.

6 Next, click the **Port** tab, and select the **Use port as input** check box. Leave the other settings at their defaults, which is `1` for **Port number** and **Fixed current density** for **Input property**.

7 Click **OK**.

MESH GENERATION

1 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.

2 Select **Coarse** mesh size from the **Predefined mesh sizes** list, and click the **Custom mesh size** option button.

3 Type 1.8 in the **Element growth rate** edit field, `0.4` in the **Mesh curvature factor** edit field, and type `0.02` in the **Mesh curvature cutoff** edit field.

4 Click **Remesh** and then click **OK**.

COMPUTING THE SOLUTION

The program can use the linear order element combination for a coarse solution and solve for the quadratic elements using the geometric multigrid solver. The linear solution is then used in the preconditioning step. COMSOL Multiphysics does this automatically with the setting selected in step 3.

1 Open the **Solver Parameters** dialog box from the **Solve** menu.

2 Click the **Settings** button. In the dialog box that appears, make sure that **Linear system solver** is selected in the field to the left. Then enter `20` in the **Factor in error estimate** edit field.

3 Select **Coarse solver** from the field. Choose **PARDISO** from the **Coarse solver** list. Set the **Tolerance** to `1.0e-5`.

4 All other settings can be left at their default values. For details on the default settings, see "Solving Large 3D Problems" on page 84 of the *AC/DC Module User's Guide*.

5 Click **OK** to close the **Linear System Solver Settings** dialog.

6 Click the **Stationary** tab, and then select **Linear** in the **Linearity** list. This model is linear, but the solver interprets the model as nonlinear, because of the coupling variable that the port condition introduces. Furthermore, with the additional dependent variable for the gauge fixing, the model has large differences in scale, which makes it difficult for the nonlinear solver to converge.

7 Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

1 Select **Plot Parameters** from the **Postprocessing** menu.

2 Make sure that the **Slice**, **Boundary**, **Arrow**, and **Geometry edges** check boxes are selected under the **General** tab.

3 Click on the **Slice** tab, and select **Magnetic flux density, norm** from the **Predefined quantities** list.

4 Type `1` in the **x levels** edit field.

5 Click on the **Boundary** tab, and select **Electric potential** from the **Predefined quantities** list.

6 Click the **Arrow** tab, and select **Magnetic flux density** from the **Predefined quantities** list.

7 In the **x points**, **y points**, and **z points** edit fields for the **Number of points**, type `20`, `7`, and `20`, respectively.

8 Select the **Cone** in the **Arrow type** list.

9 Click **OK**.

10 It is now necessary to remove some boundaries and subdomains from the plot. From the **Options** menu choose **Suppress>Boundaries**. Select all boundaries that are part of the coil. Click **Apply**, then click the **Invert Suppression** button, and finally click **Cancel**.

11 From the **Options** menu, choose **Suppress>Subdomains**. Select Subdomain 1 and click **OK**.

12 Click on the **Postprocessing mode** button, and the plot in Figure 4-19 on page 155 should appear after proper rotation and zoom operations.

13 To get the inductance value, choose **Data Display>Global** from the **Postprocessing** menu. Type `imag(Z11_emqav)/omega_emqav` in the **Expression** edit field.

14 Click **OK**.

You should get a value close to 97 μH in the message log at the bottom of the user interface.

Bonding Wires to a Chip

Electronics manufacturers use bonding wires to connect pads on an integrated circuit (IC) to the pins of the package that houses it. These bonding wires introduce relatively large inductances that can introduce crosstalk and signal distortion. This model calculates the inductance and the mutual inductance between several bonding wires.

Introduction

In today's electrical systems, high-speed communications place high demands on the packages that allow the connection of an IC to a circuit board. The most common method is to use bonding wires, which connect the pads on a semiconductor chip to the pins of a package. These bonding wires introduce relatively large inductances that can result in crosstalk and signal distortion. This model investigates the inductance of single wires and the mutual inductance between several bonding wires. The model extracts these parameters and presents them in the form of a lumped-parameter matrix, from which you can determine an impedance value. In addition, the harmonic nature of the signals results in a complex-valued matrix. Although the model extracts admittance or impedance parameters, you can also convert them to an S-parameter matrix.

Model Definition

BONDING-WIRE MODEL

Each bonding wire carries a current that results from an applied current. One end of the wire connects to the package, and the other end connects to a small pad on a chip; these pads cause a capacitive load (not included in this model). This example studies four wires at the corner of a package (Figure 4-20).

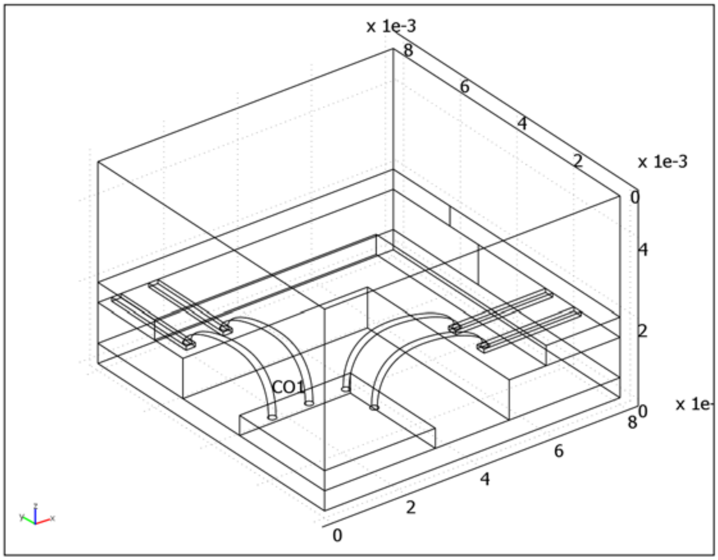


Figure 4-20: The model geometry, which studies the corner of an IC where four bonding wires connect a chip to a package.

The model tackles the problem using the Quasi-Statics, Electromagnetic application mode, where the solution variables are the electrostatic potential, V, and the magnetic vector potential, $\mathbf{A}$. Each wire has a finite conductivity, which the model handles as a separate subdomain. Because this simulation investigates effects due to the wire, it excludes any effects from vias (vertical interconnections between layers in a multilayer circuit board) and pins in the package. As a result, the model applies the potential directly to the boundary on a side of the package close to the wire, whose other side ties to the chip. The chip is modeled as a grounded plate.

LUMPED-PARAMETER CALCULATION

This model uses the *port boundary condition* with the *fixed current density* mode to model the terminals that connects each bonding wire. The fixed current density mode sets a fixed current density, so the total current gets equal to 1 A. The voltage is taken as the average over each port boundary.

The extracted voltages represent a column in the impedance matrix, $\mathbf{Z}$, which determines the relationship between the applied currents and the corresponding voltages with the formula

$$\begin{bmatrix} V_1 \\ V_2 \\ V_4 \\ V_4 \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} & Z_{13} & Z_{14} \\ Z_{21} & Z_{22} & Z_{23} & Z_{24} \\ Z_{31} & Z_{32} & Z_{33} & Z_{34} \\ Z_{41} & Z_{42} & Z_{43} & Z_{44} \end{bmatrix} \cdot \begin{bmatrix} I_1 \\ I_2 \\ I_3 \\ I_4 \end{bmatrix} .$$

Thus, when $I_1 = 1$ and all the other currents are zero, the **V** vector equals the first column of **Z**. You get the full matrix by setting each of the currents to one. With this matrix, you can calculate other types of parameter matrices. For more information on lumped parameter calculations in the AC/DC Module, see "Lumped Parameters" on page 61 in the *AC/DC Module User's Guide*.

To calculate the entire matrix, you must export the model's entire FEM structure to the COMSOL Script command line. A function called `rlcmatrix` performs the calculation, which solves all the necessary port conditions to extract the parameter matrix. See the function `rlcmatrix` on page 117 in the on-line manual *AC/DC Module Reference Guide* for more information.

Results and Discussion

The inductive effects clearly dominate in this model. At a frequency of 10 MHz, the inductance matrix is

$$\mathbf{L} = \begin{bmatrix} 2.6 & 0.59 & 0.0052 & 0.0039 \\ 0.59 & 3.5 & 0.029 & 0.015 \\ 0.0052 & 0.029 & 3.7 & 0.96 \\ 0.0039 & 0.015 & 0.96 & 3.5 \end{bmatrix} \text{(in nH)}$$

where the model calculates **L** from the imaginary part of the impedance matrix, **Z**, using the relationship $\mathrm{imag}(\mathbf{Z}) = \omega\mathbf{L}$. The **L** matrix is symmetric, which is a property most lumped parameter matrices have.

It is always important to check the extracted **Z** matrix before analyzing the other parameters, especially the ratio between the real and complex parts, and the condition number of the matrix. Furthermore, the boundary conditions create mirror wires outside the simulation domain, and they influence the extracted parameters.

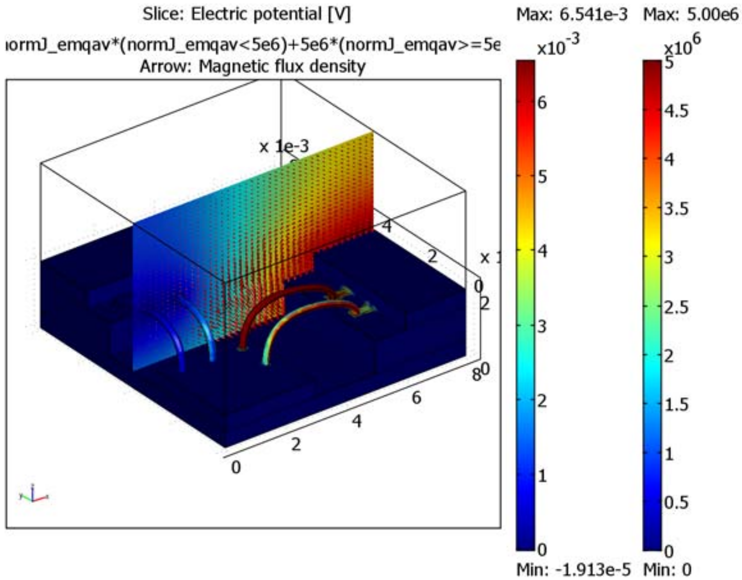


Figure 4-21: This plot shows a slice plot, an arrow plot of the magnetic flux density, and a boundary plot of the current density. For this solution, the third bonding wire is used as input, giving the third column in the parameter matrix.

Model Library path: `ACDC_Module/Electrical_Components/bond_wires_to_chip`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator**, select **3D** in the **Space dimension** list.
2 In the **AC/DC Module>Quasi-Statics, Electromagnetic** folder, select the **Electric and Induction Currents** application mode.
3 Select **Vector - Linear** from the **Element** list.
4 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Constants**.

2 In the **Constants** dialog box, define the following constants with names, expressions, and descriptions (optional):

NAME	EXPRESSION	DESCRIPTION
`epsilonr_pcb`	`2.5`	Relative permittivity of package material
`sigma_pcb`	`10`	Conductivity of package material (S/m)

3 Click **OK**.

GEOMETRY MODELING

1 From the **File** menu, select **Import>CAD Data From File**.

2 In the **Import CAD Data From File** dialog box, make sure that the **COMSOL native format** or **All 3D CAD files** is selected in the **Files of type** list.

3 Locate the `bond_wires_to_chip.mphbin` file, and click **Import**.

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

PHYSICS SETTINGS

Scalar Variables

1 From the **Physics** menu, choose **Scalar Variables**.

2 In the **Scalar Variables** dialog box, change the value of the frequency, `nu_emqav`, to `10e6`.

3 Click **OK**.

Subdomain Settings

1 From the **Physics** menu, open the **Subdomain Settings** dialog box.

2 Define the subdomain settings according to the following table:

SETTINGS	SUBDOMAINS 1, 4, 5	SUBDOMAIN 2	SUBDOMAINS 6–13
Material		Silicon	Aluminum
e_r	`epsilonr_pcb`		
σ	`sigma_pcb`		

3 Click **OK**.

Boundary Conditions

1 From the **Physics** menu, open the **Boundary Settings** dialog box.

2 Select all boundaries, click the **Electrical Parameters** tab, and select the **Electric insulation** boundary condition from the **Boundary Condition** list.

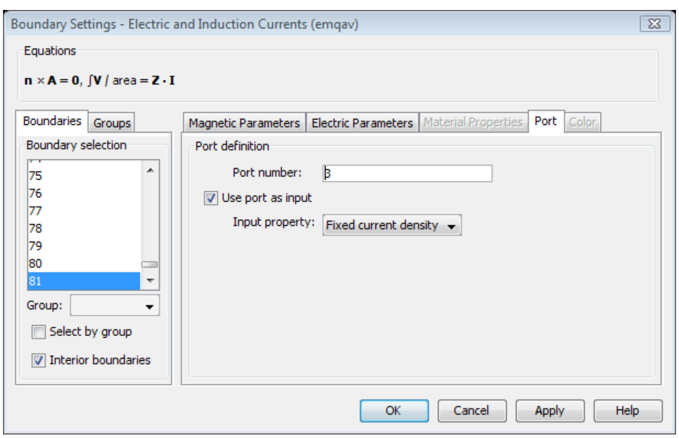
3 Select the **Interior boundaries** check box to allow settings on interior boundaries.

4 Enter the other boundary settings according the following two tables (leave all fields not specified at their default values):

SETTINGS	BOUNDARY 30	BOUNDARY 42
Electric boundary condition	Port	Port
Port number	1	2
Use port as input	cleared	cleared
Input property	Fixed current density	Fixed current density

SETTINGS	BOUNDARY 81	BOUNDARY 80	BOUNDARIES 33, 45, 50, 52
Magnetic boundary condition	Magnetic insulation	Magnetic insulation	Magnetic insulation
Electric boundary condition	Port	Port	Ground
Port number	3	4	
Use port as input	selected	cleared	
Input property	Fixed current density	Fixed current density	

5 Click **OK**.



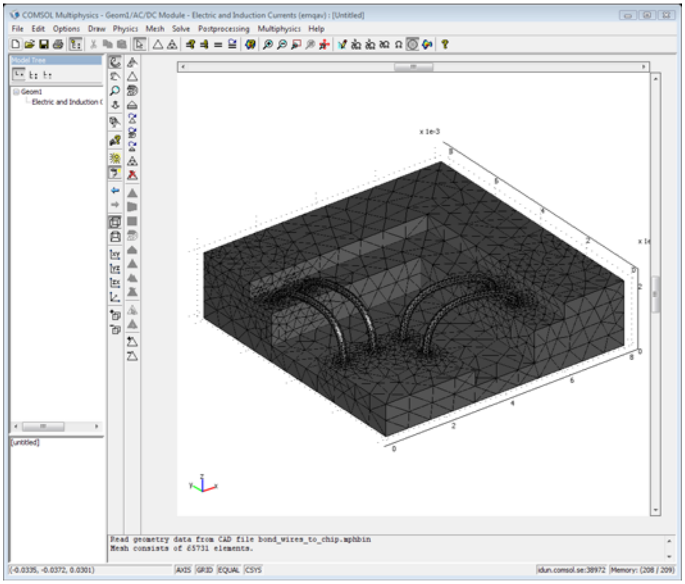
MESH GENERATION

1 From the **Mesh** menu choose **Free Mesh Parameters**.

2 In the **Element growth rate** edit field type `1.4`.

3 Click the **Remesh** button, then click **OK**.

4 From the **Options** menu select **Suppress>Suppress Boundaries**. Select Boundaries 7, 8, 10, 24, and 79. Click **OK**. Click the **Mesh mode** button to see the following figure.



COMPUTING THE SOLUTION

1 From the **Solve** menu, choose **Solver Parameters**.

2 From the **Linear solver** list, select the **GMRES** solver.

3 From the **Preconditioner** list, select **Vanka**.

4 Click the **Settings** button.

5 In the **Linear System Settings** dialog box, enter `10` in the **Factor in error estimate** edit field and `200` in the **Number of iterations before restart** edit field.

6 Select **Preconditioner** for the tree view, and enter 3 in the **Number of iterations** edit field.

7 Click **OK**.

8 Click **OK** to close the **Solver Parameters** dialog box.

9 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 From the **Postprocessing** menu select **Plot Parameters**.
2 Make sure that the **Slice**, **Boundary**, **Arrow**, and **Geometry edges** check boxes are selected under the **General** tab.
3 Click the **Slice** tab, then select **Magnetic flux density, norm** from the **Predefined quantities** list.
4 Type `0` in the **x levels** edit field. Click the **Vector with coordinates** option button for the **y levels**. Enter `4e-3` in the corresponding edit field.
5 Click the **Boundary** tab, and type `nJ_emqav*(nJ_emqav<5e6)+5e6*(nJ_emqav>=5e6)` in the **Expression** edit field.
6 Click the **Range** button. Clear the **Auto** check box and type `0` and `5e6` in the **Min** and **Max** edit fields, respectively. Click **OK**.
7 Click the **Arrow** tab, then select **Magnetic flux density** from the **Predefined quantities** list.
8 Click the **Vector with coordinates** option button for the **y points**. Type the coordinate `3.9e-3` in the edit field next to that option button.
9 In the **x points** and **z points** edit fields for the **Number of points** type `40`.
10 Select the **3D arrow** in the **Arrow type** list.
11 Click the **Color** button, then select the white color from the palette. Click **OK**.
12 Click **OK** in the **Plot Parameters** dialog box to get the plot in Figure 4-21 on page 164.

Modeling Using the Programming Language

Note: This section requires COMSOL Script or MATLAB.

PARAMETER-MATRIX CALCULATION

The `rlcmatrix` function goes through all the port boundary conditions, setting the **Use port as input** property on one port at a time, and solves the problem. To execute this script, you first need to export the FEM structure to the COMSOL Script command line. Do the following steps to get the full **Z** lumped parameter matrix. Note that this requires a 64-bit platform, and that the calculation of the matrix takes about 2 hours.

1 Export the FEM structure by pressing Ctrl+F. The COMSOL Script window opens automatically.

2 Enter the following at the command prompt:

```
Z = rlcmatrix(fem,'Z');
freq = postint(fem,'nu_emqav','edim',0,'dl',1);
L = imag(Z)/(2*pi*freq)*1e9
```

The output reads

```
L =

     2.6482     0.5944     0.0052     0.0039
     0.5944     3.4833     0.0285     0.0149
     0.0052     0.0285     3.6593     0.9607
     0.0039     0.0149     0.9607     3.4919
```

This is the inductance matrix given in nH presented in “Results and Discussion” on page 163.

High Current Cables in a Circuit

This model connects two high current cables to a battery in a battery charger circuit.

Introduction

When charging a car battery, a significant amount of current can pass in the cables from the charger to the battery. The cable has two sensitive areas that cause additional voltage drop; the junction between the copper wire and the clip, and the contact junction at the battery pole. See Figure 4-22 for a complete view over the system. At high currents these junctions produce a lot of heat, which can melt the cables if the junctions suffer from high contact resistances. The contact resistance can degrade in corrosive enviorments.

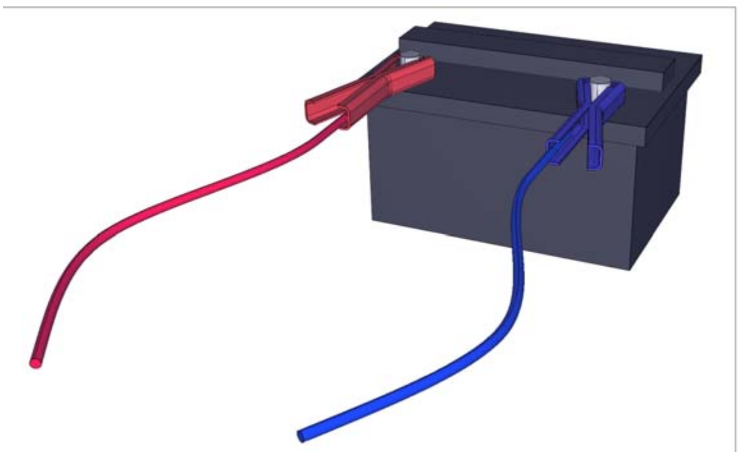


Figure 4-22: A view of the entire system, where the cables, clips, and poles are modeled in a physical model. All other components are modeled as circuit models.

Model Definition

This model uses simple circuit models for the charger and the battery. A physical model in a separate model file contains the copper wire, clip and battery pole. Because the two cables between the battery and charger are identical, two instances of this separate model file represent the cables. The circuit models and the two identical physical models then represent the entire system.

The circuit model of the charger is simply a voltage source with a resistor in series. The resistor has a low value, so a significant amount of current can be delivered from the charger. The battery circuit model is also a voltage source but without the series resistor, because it can be neglected in comparison to the series resistor of the charger. As mentioned previously, the cables originate from the same COMSOL Multiphysics model, implementing two subcircuit instances in the circuit. The cable on the minus side of the battery also has a resistor in parallel, representing the alternative current paths to ground.

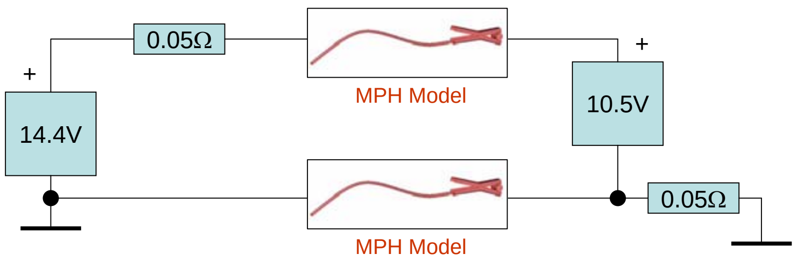


Figure 4-23: A block diagram over the entire charger system.

The physical model of the cable uses an imported geometry created in SolidWorks®, which was directly imported to COMSOL Multiphysics using the SolidWorks direct connection. After importing the geometry, it was exported to the COMSOL native format (an mphbin-file) for easier access. The cable geometry consists of a copper cable, a clip, and a battery pole, which the software treats as separate geometry objects to allow assembly modeling. Assemblies are necessary to model the relatively high contact resistance at the junctions between the different objects, because assemblies allow a discontinuity in the potential.

The contact resistance depends on several properties of the junction, like contact pressure, ambient conditions, and surface oxides, so it is hard to find exact values of the quantity. The values used here are just coarse approximations, and summarized in Table 4-1 on page 172. You must provide the model parameters for the contact resistance pair conditions as an equivalent interfacial layer, where the layer thickness has no resemblance to reality.

TABLE 4-1: MODEL PARAMETERS FOR THE CONTACT RESISTANCES

JUNCTION	**CONTACT RESISTANCE**	**MODEL PARAMETERS**
cable-clip	100 μΩcm	$\sigma = 10^6$ S/m, d = 1 m
clip-pole	10 μΩcm	$\sigma = 10^7$ S/m, d = 1 m

The boundaries connecting to the circuit use a boundary condition called Circuit terminal, which is specialized for circuit connections. The SPICE import automatically finds the circuit terminals defined in the model and connect them to the specified nodes in the circuit for a particular instance. All other boundaries use the electric insulation boundary condition.

The wire and clip are made of copper, and the battery pole is made of a lead alloy. The only lead alloy present in the basic material library is a solder material, so to simplify the modeling this material is used as the lead alloy in the battery pole. All relevant material properties are summarized in the table below.

TABLE 4-2: MATERIAL PARAMETERS

MATERIAL	**RESISTIVITY**	**TEMPERATURE COEFFICIENT**	**THERMAL CONDUCTIVITY**
Copper	17.8 nΩm	0.0039 K^{-1}	400 K/(m·W)
Solder, 60Sn-40Pb	499 nΩm	0.0 K^{-1}	50 K/(m·W)

The heat generated at the junctions has to be accounted for. Using the pair condition called Heat flux discontinuity, you can specify a heat source that is equal to the formula

$$q_0 = d\mathbf{E}\cdot\mathbf{J} = d\frac{V_{\mathrm{src}}-V_{\mathrm{dst}}}{d}\sigma(V_{\mathrm{src}}-V_{\mathrm{dst}}) = \sigma(V_{\mathrm{src}}-V_{\mathrm{dst}})^2$$

where V_{src} is the voltage at the source side of the pair, and V_{dst} the voltage at the destination side. The variable `Qs_emdc` returns the value of the above formula if it is used on the boundaries of a contact resistance pair condition. The boundaries attached to the charger and battery are fixed at room temperature. All other boundaries have thermal insulation, so the dissipation to air is neglected.

Results and Discussion

When the cables are part of the charger circuit the current flowing in the cables are different. An electrical system using car batteries usually has the minus side of the battery connected to all metal parts, which represents the alternative path to ground in the circuit. This results in a larger current in the cable connected to the plus side of the battery. The following figures show the potential distribution of both cables.

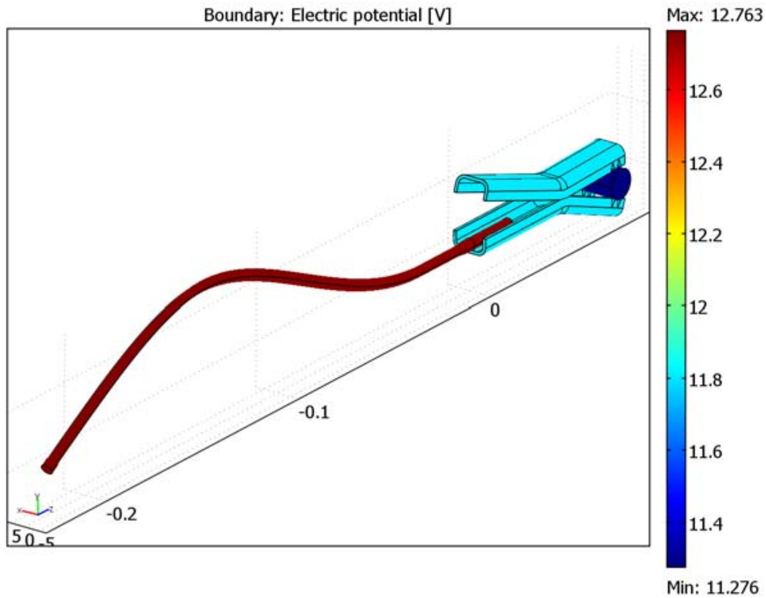


Figure 4-24: The potential distribution of the cable on the plus side of the battery. Almost the entire voltage drop is over the two junctions.

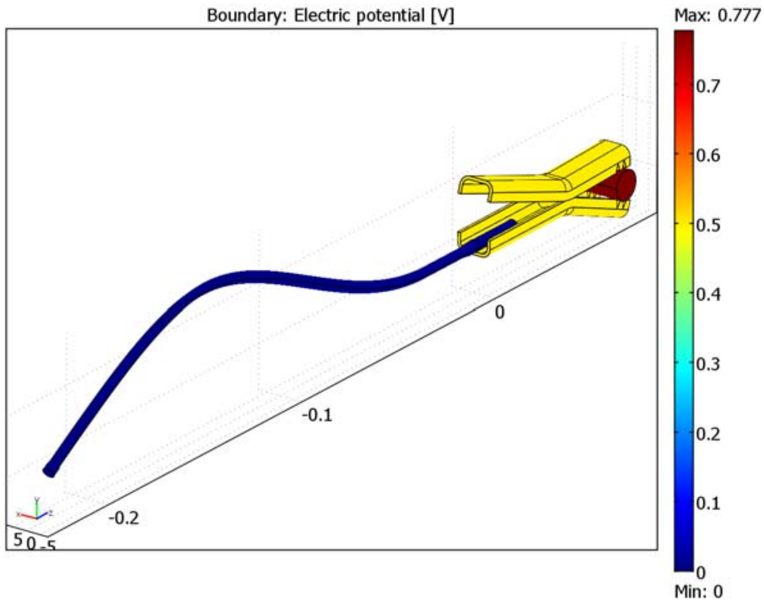


Figure 4-25: The potential distribution of the cable on the minus side of the battery.

Because most of the voltage drop is over the junctions, the major part of the heat generation is also located there. A temperature plot of both cables confirms this.

Especially the cable-clip junction is sensitive to heat if the cable is soldered to the clip (common on less expensive cables) or if the junction has suffered from corrosion.

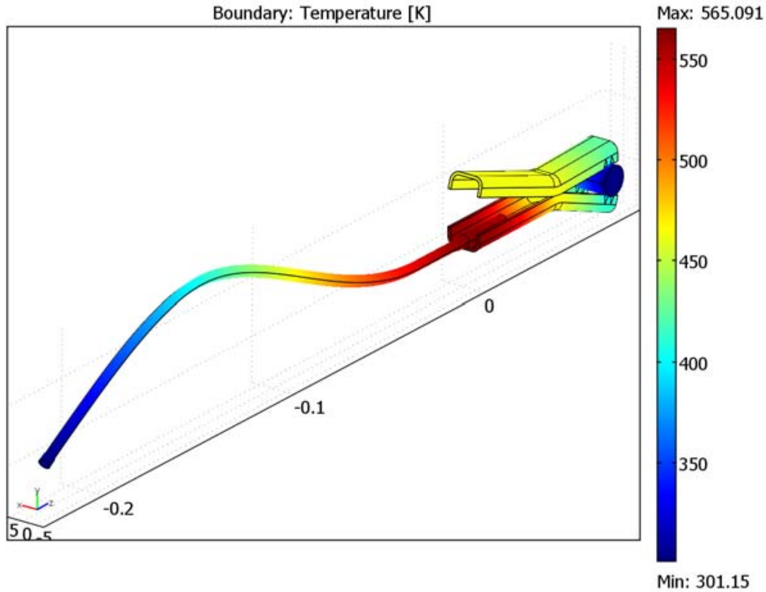


Figure 4-26: The temperature distribution on the plus side of the battery. The area around the cable-clip junction gets a high temperature that can damage a soldered junction.

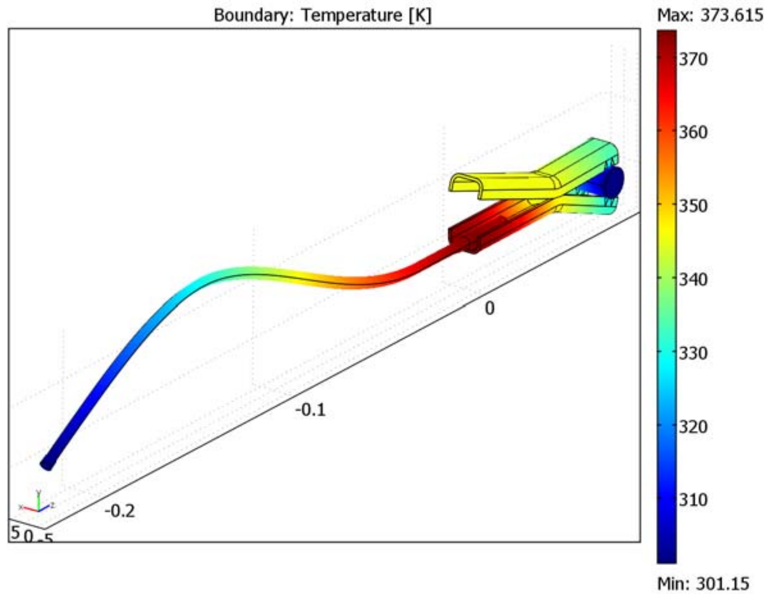


Figure 4-27: The temperature distribution on the minus side of the battery. The distribution is less critical here due to the alternative path for the current.

Model Library path: `ACDC_Module/Electrical_Components/high_current_cables_circuit`

Modeling Using the Graphical User Interface

The modeling steps below assume that the Heat Transfer Module is not present. If it is, the Joule Heating multiphysics node uses the Heat Transfer Module's version of the Heat Transfer by Conduction application mode. As a result, some names referring to components in the graphical user interface are slightly different.

MODEL NAVIGATOR

1. In the **Model Navigator**, select **3D** from the **Space dimension** list.
2. In the **AC/DC Module>Electro-Thermal Interaction** folder, select the **Joule Heating** multiphysics node.

3 Click **OK**.

GEOMETRY MODELING

1 From the **File** menu, select **Import>CAD Data From File**.

2 In the **Import CAD Data From File** dialog box, make sure that the **COMSOL native format** or **All 3D CAD files** is selected in the **Files of type** list.

3 Locate the `high_currect_cable.mphbin` file, and click **Import**.

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

5 Select all objects by pressing Ctrl+A.

6 Click the **Create Pairs and Imprints** button on the Draw toolbar.

PHYSICS SETTINGS (CONDUCTIVE MEDIA DC)

Subdomain Settings

1 From the **Physics** menu, open the **Subdomain Settings** dialog box.

2 Select subdomains 2, 3, and 4, then click the **Load** button.

3 In the **Materials/Coefficients Library** dialog box, expand the **Basic Material Properties** library and choose **Copper**.

4 Click **OK** to close the dialog box.

5 From the **Conductivity relation** list, choose **Linear temperature relation**.

6 Select subdomain 1 and click the **Load** button.

7 In the **Materials/Coefficients Library** dialog box, expand the **Basic Material Properties** library and choose **Solder, 60Sn-40Pb**.

8 Click **OK** to close the dialog box.

9 From the **Conductivity relation** list, choose **Linear temperature relation**.

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

Boundary Conditions

1 From the **Physics** menu, open the **Boundary Settings** dialog box.

2 Select all boundaries and choose the **Electric insulation** boundary condition from the **Boundary condition** list.

3 Select Boundary 1 and choose **Ground** from the **Boundary condition** list.

4 Select Boundary 163, choose **Electric potential**, and enter `1` in the **Electric potential** edit field.

5 Click the **Pairs** tab and select **Pair 1 (identity)** from the **Pair selection** area.

6 From the **Boundary condition** list, choose **Contact resistance** and enter `1e7` in the **Electric conductivity** edit field.

7 Select the second pair, **Pair 2 (identity)**, choose **Contact resistance** and enter `1e6` in the **Electric conductivity** edit field.

8 Click **OK**.

PHYSICS SETTINGS (HEAT TRANSFER BY CONDUCTION)

Subdomain Settings

1 From the **Model Tree** browser, double-click on the item **Geom1>Heat Transfer by Conduction>Subdomain Settings**. This opens the **Subdomain Settings** dialog box for the Heat Transfer application mode.

2 In the dialog box, select subdomains 2, 3 and 4, then choose **Copper** from the **Library material** list.

3 Select Subdomain 1, and choose **Solder, 60Sn-40Pb** from the **Library material** list.

4 Click the **Init** tab, select all subdomains, and enter `28[degC]` in the **Temperature** edit field. The unit `degC` specifies the unit to degrees Celsius.

5 Click **OK**.

Boundary Conditions

1 From the **Model Tree** browser, double-click on the item **Geom1>Heat Transfer by Conduction>Boundary Settings**.

2 In the **Boundary Settings** dialog box, select Boundaries 1 and 163, and choose **Temperature** from the **Boundary condition** list.

3 Enter `28[degC]` in the **Temperature** edit field.

4 Click the **Pairs** tab, select both pairs and choose **Heat flux discontinuity** from the **Boundary condition** list.

5 Enter `Qs_emdc` in the **Inward heat flux** edit field.

6 Click **OK**.

MESH GENERATION

1 From the **Mesh** menu choose **Free Mesh Parameters**.

2 Click the **Boundary** tab, and select Boundaries 103, 104, 167, and 168. Then enter `1e-3` in the **Maximum element size** edit field.

3 Click the **Remesh** button, then click **OK**.

COMPUTING THE SOLUTION

This is a problem that benefits from a segregated solver approach, both regarding stability and memory consumption.

1 From the **Solve** menu, choose **Solver Parameters**.
2 In the **Solver Parameters** dialog box, select the **Stationary segregated** solver.
3 Click the **Settings** button for group number 1.
4 In the **Linear System Solver Settings** dialog box, choose **Direct (SPOOLES)** from the **Linear system solver** list.
5 Click **OK**.
6 Click the **Settings** button for group number 2, and repeat step 4-5 for this group.
7 Select the **Manual specification of segregated steps** check box, and specify the steps according the following table.

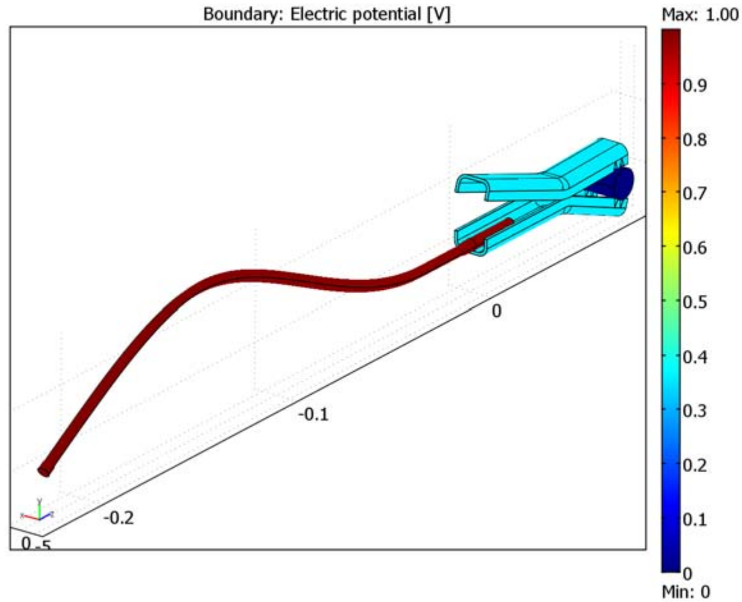
GROUP	DAMPING	NUMBER OF ITERATIONS
2	0.8	1
1	1	2

8 Click **OK**.
9 Click the **Solve** button on the Main toolbar.

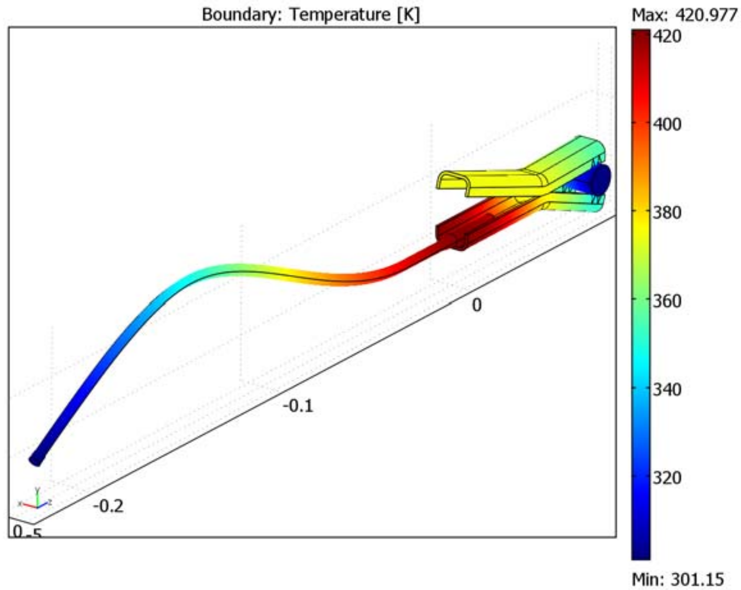
POSTPROCESSING AND VISUALIZATION

1 From the **Postprocessing** menu, choose **Plot Parameters**.
2 In the **Plot Parameters** dialog box, clear the **Slice** check box, and select the **Boundary** check box.
3 Click the **Boundary** tab, and choose **Conductive Media DC (emdc)>Electric potential** from the **Predefined quantities** list.

4 Click **Apply** to see the following plot.



5 To plot the temperature, choose **Heat Transfer by Convection (ht)>Temperature** for the **Predefined quantities** list, and click **OK**.



PREPARING FOR CIRCUIT CONNECTION

The following steps are a preparation for the SPICE connection, which has to be done before the model is used in a circuit.

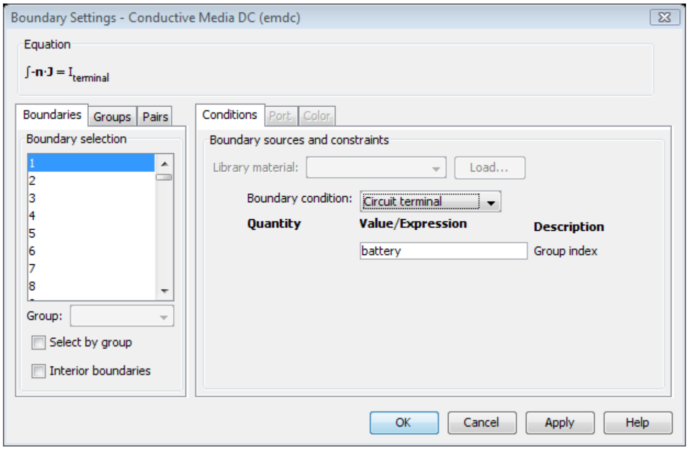
Boundary Conditions

1 From the **Model Tree** browser, double-click on the item **Geom1>Conductive Media DC (emdc)>Boundary Settings**.

2 In the **Boundary Settings** dialog box, select Boundary 163 and choose **Circuit terminal** from the **Boundary condition** list.

3 Enter `charger` in the **Group index** edit field.

4 Select Boundary 1 and choose **Circuit terminal**, then enter `battery` in the **Group index** edit field.



5 Click **OK**.

Save the Model

From the **File** menu, choose **Save As**, then choose a proper location for the file. The file name should be `high_current_cable.mph` to be compatible with the modeling steps in the next section.

Modeling the Entire Circuit

The following steps assume that you continue from the previous section.

MODEL NAVIGATOR

1 From the **File** menu, choose **New**.

2 In the **Model Navigator** dialog box, click **OK**.

SPICE IMPORT

1 From the **Physics** menu, choose **SPICE Circuit Editor**.

2 In the **SPICE Circuit Editor** dialog box, click the **Create Voltage Source** toolbar button.

3 In the **Create Voltage Source** dialog box, enter `V1` in the **Device instance name** edit field.

4 Then enter `1 0` in the **Terminal names** edit field, and `14.4V` in the **Device value** edit field.

5 Click **OK**. You should now see the following text in the text area: `V1 1 0 14.4V`.

6 Click the **Create Resistor** toolbar button.

7 In the dialog box that appears, enter `R1` in the **Device instance name** edit field, `1 2` in the **Terminal names** edit field, and `0.05` in the **Device value** edit field.

8 Click **OK**. This adds a statement representing a resistor between nodes 1 and 2 with a value of 0.05 Ω.

9 Click the **Create Subcircuit Instance** toolbar button, enter `X1` in the **Device instance name** edit field, `2 3` in the **Terminal names** edit field, and `cable` in the **Subcircuit reference name** edit field.

10 Click **OK**. This adds a statement representing a subcircuit instance of the subcircuit `cable` between nodes 2 and 3.

11 Click the **Create Voltage Source** toolbar button, enter `Vbattery` in the **Device instance name** edit field, `3 4` in the **Terminal names** edit field, and `10.5V` in the **Device value** edit field.

12 Click **OK**. This adds the voltage source representing the battery.

13 Click the **Create Subcircuit Instance** toolbar button, enter `X2` in the **Device instance name** edit field, `0 4` in the **Terminal names** edit field, and `cable` in the **Subcircuit reference name** edit field.

14 Click **OK**. This statement adds a second subcircuit instance of the same subcircuit as `X1`.

15 Click the **Create Resistor** toolbar button, enter `R2` in the **Device instance name** edit field, `4 0` in the **Terminal names** edit field, and `0.05` in the **Device value** edit field.

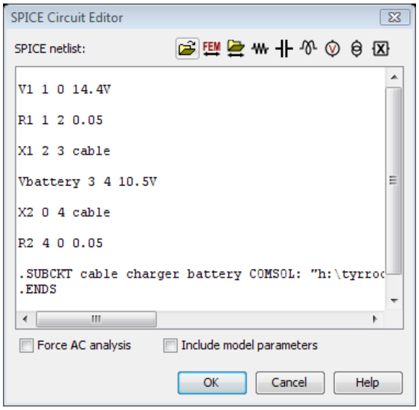
16 Click **OK**. This statement represents the alternative paths to ground in the system.

17 Click the **Create Link to Model File** toolbar button, enter `cable` in the **Subcircuit reference name** edit field, and `charger battery` in the **Terminal names** edit field.

18 Click the **Select File** button.

19 In the dialog box that appears, browse to the file named `high_current_cable.mph` you saved in the previous section. As an alternative, you can also browse to the Model Library version of the file with the same name located in the Model Library path of this model.

20 Click the **Create Link** button. The entire netlist should look similar to the figure below.



21 Click **OK** to close the **SPICE Circuit Editor** dialog box. COMSOL Multiphysics now adds all the global variables and ODE expression that represents the netlist. For the subcircuit instances, it adds the model in the model file twice as two new geometries, using the add component feature.

COMPUTING THE SOLUTION

This is a problem that can benefit from a segregated solver approach, both regarding stability and memory consumption. Scaling of the circuit variables can sometimes be an issue, so it is also necessary to scale them manually. You set the scaling of these variables an order of magnitude lower than the expected value to increase the weight of them in the solution of the problem.

1 From the **Solve** menu, choose **Solver Parameters**.

2 In the **Solver Parameters** dialog box, select the **Stationary segregated** solver.

3 Click the **Settings** button for group number 1.

4 In the **Linear System Solver Settings** dialog box, choose **Direct (SPOOLES)** from the **Linear system solver** list.

5 Click **OK**.

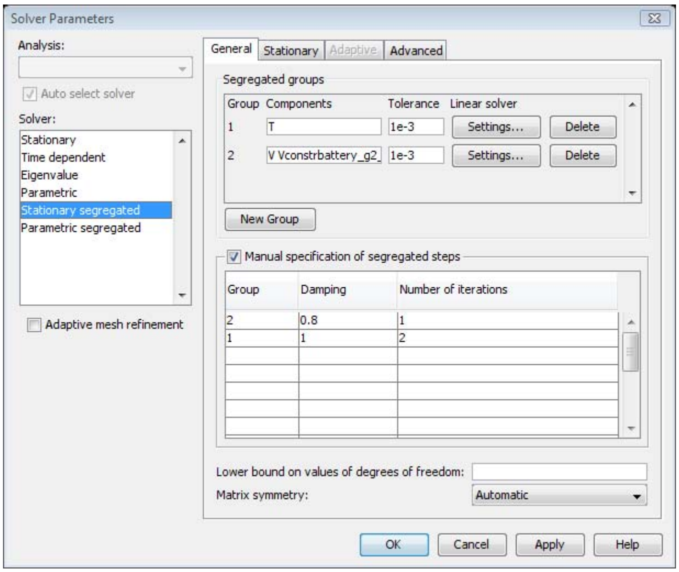
6 In the **Components** edit field of group number 2, enter all solution variables except the temperature variable T. The list below shows the variables to enter, but you enter them as a space-separated list in the edit field.

SOLUTION VARIABLE
V
Vconstrbattery_g2_emdc
Vconstrcharger_g2_emdc
Vconstrbattery_g3_emdc
Vconstrcharger_g3_emdc
I0_X2_cir
I2_X1_cir
I3_X1_cir
I4_X2_cir
I_V1_cir
I_VBATTERY_cir
V1_cir

7 Select the **Manual specification of segregated steps** check box, and specify the steps according the following table.

GROUP	DAMPING	NUMBER OF ITERATIONS
2	0.8	1
1	1	2

The dialog box at this step is shown below.



8 Click the **Advanced** tab.

9 From the **Type of scaling** list, choose **Manual**.

10 Enter a space-separated list containing all solution variables followed by its scale. The entire solution variable-value pair is listed in the table below.

SOLUTION VARIABLE	SCALE VALUE
T	300
V	10
Vconstrbattery_g2_emdc	0.1
Vconstrcharger_g2_emdc	0.1
Vconstrbattery_g3_emdc	0.1
Vconstrcharger_g3_emdc	0.1
I0_X2_cir	0.1
I2_X1_cir	0.1
I3_X1_cir	0.1
I4_X2_cir	0.1
I_V1_cir	0.1

The beginning of the space-separated list will then be

```
T 300 V 10 Vconstrbattery_g2_emdc 0.1 Vconstrcharger_g2_emdc ...
```

11 Click **OK** to close the **Solver Parameters** dialog box.

12 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

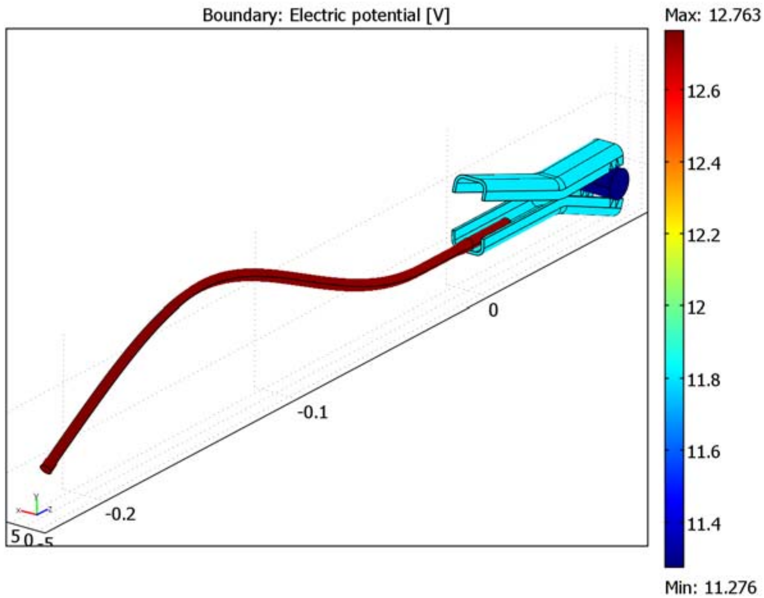
1 Click the **Geom2** tab in the main user interface.

2 From the **Postprocessing** menu, choose **Plot Parameters**.

3 In the **Plot Parameters** dialog box, clear the **Slice** check box, and select the **Boundary** check box.

4 Click the **Boundary** tab, and choose **Conductive Media DC (emdc)>Electric potential** from the **Predefined quantities** list.

5 Click **Apply** to see the following plot, which is also shown in the section “Results and Discussion” on page 172.



6 Repeat these steps for the temperature and for both variables in the last geometry to get the other plots in the “Results and Discussion” section.

Inductor in Amplifier Circuit

This model studies a finite element inductor inserted in an electrical amplifier circuit.

Introduction

Modern electronic systems are very complex and depend heavily on computer aided design in the development and manufacturing process. Common tools for such calculations are based on the SPICE format originally developed at Berkeley University (Ref. 1). The SPICE format consists of a standardized set of models for describing electrical devices—especially semiconductor devices such as transistors, diodes, and thyristors. SPICE also includes a simple, easy-to-read text format for circuit netlists and model parameter specifications. Although the netlist format is essentially the same as it was from the beginning, the set of models and model parameters are constantly changing, with new models being added according to the latest achievements in semiconductor device development. When the devices are scaled down, new effects appear that have to be properly modeled. The new models are the result of ongoing research in device modeling.

When an engineer is designing a new electronic component, like a capacitor or an inductor, the SPICE parameters for that device are not known. They are either extracted from finite element tools, such as COMSOL Multiphysics, or from measurements on a prototype. To speed up the design process it can be convenient to include the finite element model in the SPICE circuit simulation, calculating the device behavior in an actual circuit.

This model takes a simple amplifier circuit and exchanges one of its components with a finite element model of an inductor with a magnetic core. COMSOL Multiphysics calculates the transient behavior of the entire system. A script adds the circuit elements as ODE equations to the inductor model along with the necessary model parameters for the SPICE devices in the circuit.

Model Definition

The inductor model uses the Azimuthal induction currents application mode of the AC/DC Module, solving for the magnetic potential **A**.

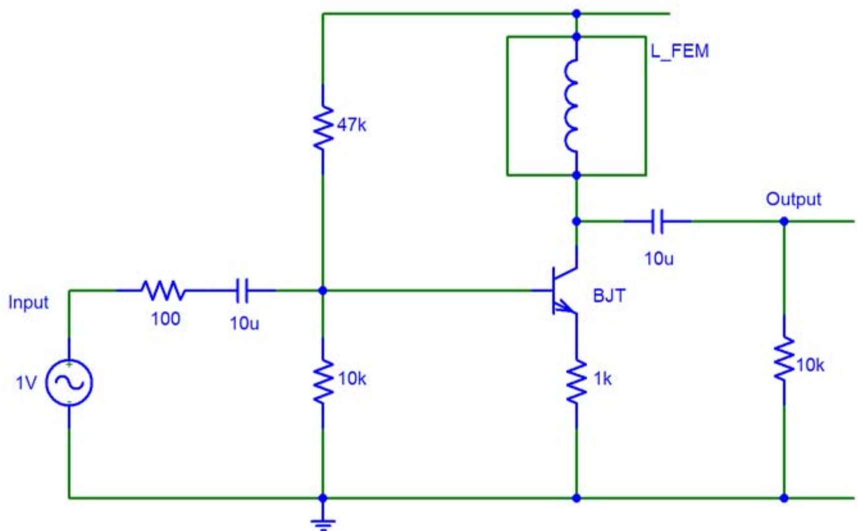
$$\sigma\frac{\partial \mathbf{A}}{\partial t} + \nabla \times (\mu_0^{-1}\mu_r^{-1}\nabla \times \mathbf{A}) = \mathbf{J}_e$$

where μ_0 is the permeability of vacuum, μ_r the relative permeability, and σ the electric conductivity.

Because the inductor has a large number of turns it is not efficient to model each turn as a separate wire. Instead the entire coil is treated as a block with a constant external current density corresponding to the current in each wire. The conductivity in this block is zero to avoid eddy currents, which is motivated with the fact that no currents can flow between the individual wires. The eddy currents within each wire is neglected.

CONNECTION TO A SPICE CIRCUIT

The electrical circuit is a standard amplifier circuit with one bipolar transistor, biasing resistors, input filter, and output filter, see the figure below.



The input is a sine signal of 1 V and 10 kHz. The SPICE netlist for this circuit is shown below.

```
* BJT Amplifier circuit
.OPTIONS TNOM=27
.TEMP 27
Vin    1    0    sin(0 1 10kHz)
Vcc    4    0    15
Rg     1    2    100
Cin    2    3    10u
R1     4    3    47k
R2     3    0    10k
X1     4    5    inductor
RE     7    0    1k
```

```
Cout   5    6    10u
Rl     6    0    10k
Q1     5    3    7    BJT
.MODEL BJT  NPN(Is=15f Ise=15f Isc=0 Bf=260 Br=6.1
+ Ikf=.3 Xtb=1.5 Ne=1.3 Nc=2 Rc=1  Rb=10 Eg=1.11
+ Cjc=7.5p Mjc=.35 Vjc=.75 Fc=.5 Cje=20p Mje=0.4 Vje=0.75
+ Vaf=75 Xtf=3 Xti=3)
.SUBCKT inductor V_coil I_coil COMSOL: *
.ENDS
.END
```

The device X1 refers to a subcircuit defined at the end of the file. The subcircuit definition is part of the SPICE standard to define blocks of circuits that can be reused in the main circuit. The special implementation used here defines a subcircuit that really is a Comsol Multiphysics model, referenced with the option `COMSOL: <file name> | <application mode name> | *`. The asterisk means that COMSOL Multiphysics looks for the first occurrence of the specified parameters `V_coil` and `I_coil` in the current model. These parameters are the variables that links the model with the circuits, and must be defined in the model in a certain way. The variable `V_coil` must give the voltage over the device, defined in the global scope. `I_coil` must be a global variable used in the model as a current through the device.

The script can also handle model files with more than two terminals. It is then necessary to define each terminal as a floating-potential boundary condition. Note that any use of the ground boundary condition will connect those boundaries directly to the ground node of the circuit, labeled `0`.

The model parameters of the transistor do not correspond to a real device, but the numbers are nevertheless chosen so as to be realistic.

The import of the SPICE netlist does not fully support the SPICE format; especially for the semiconductor device models, it only supports a limited set of parameters. Supplying unsupported parameters does not give an error message, but the parameter is not used in the circuit model. For example, the transit time capacitance and the temperature dependence are not supported for the transistor model.

Results and Discussion

Biasing of an amplifier is often a complicated compromise, especially if you only use resistors. Adding an inductor as the collector impedance simplifies the biasing design, since the instantaneous voltage on the collector of the transistor can be higher than the supply voltage, which is not possible with resistors. Amplifiers using inductors can be quite narrow banded.

Before starting the transient simulation, proper initial conditions have to be calculated. For this model it is sufficient to ramp the supply voltage to 15 V with the nonlinear parametric solver. After the ramp, the DC bias conditions have been calculated properly and this solution can be used as initial condition for the transient simulation. After 0.2 ms the magnetic field and contour lines of the vector potential should look like Figure 4-28 below.

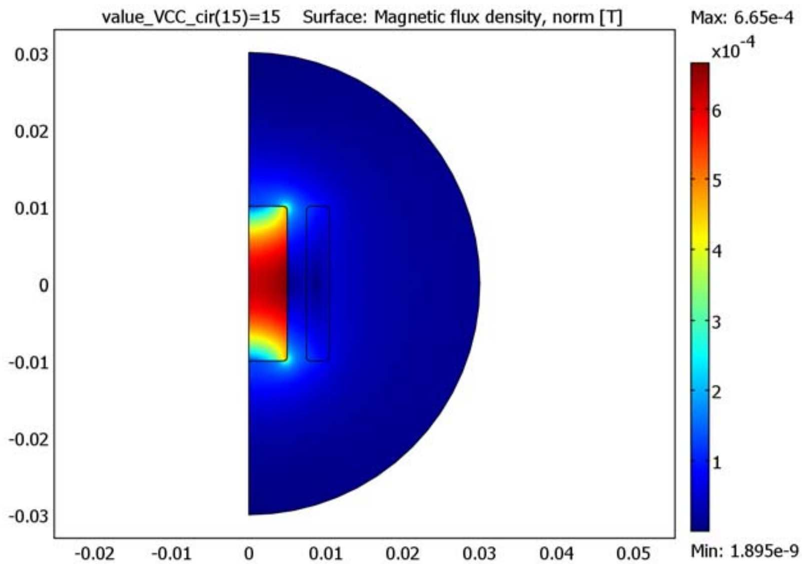


Figure 4-28: Magnetic flux density (color) after the bias point calculation.

Using the global variable plot, input signal, output signal, and inductor voltage can easily be plotted in the same graph.

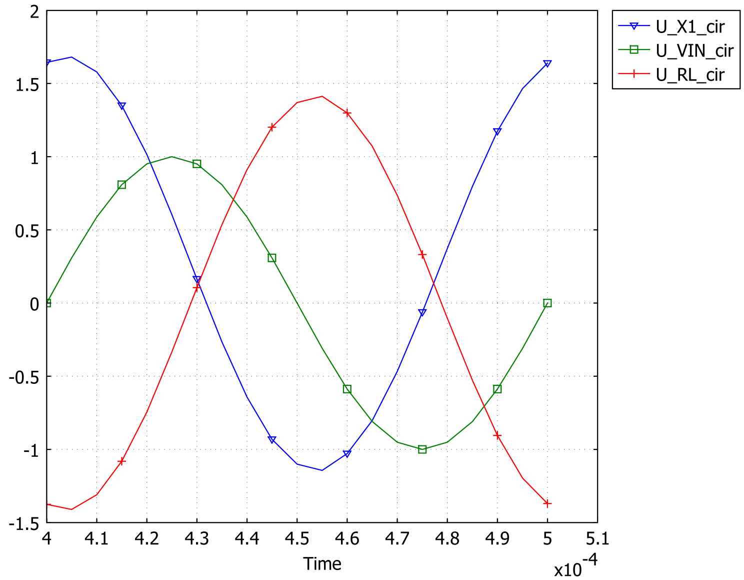


Figure 4-29: Input signal (`U_VIN_cir`), output signal (`U_RL_cir`), and inductor voltage (`U_X1_cir`) as a function of time.

The output signal is about 1.5 times the input signal in amplitude.

Reference

1. "The SPICE home page," http://bwrc.eecs.berkeley.edu/Classes/IcBook/SPICE.

Model Library path: `ACDC_Module/Electrical_Components/amplifier_and_inductor`

MODEL NAVIGATOR

1 In the **Model Navigator**, select **Axial symmetry (2D)** from the **Space dimension** list.

2 In the **AC/DC Module** folder, select **Statics>Magnetostatics>Azimuthal Induction Currents, Vector Potential**.

3 Click **OK** to close the **Model Navigator**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Constants**.

1 In the **Constants** dialog box, define the following constants with names and expressions. The description field is optional.

NAME	EXPRESSION	DESCRIPTION
`t`	`0[s]`	`Time for stationary solution`
`N`	`1e3`	`Turns in coil`
`freq`	`10[kHz]`	`Frequency`
`r_coil`	`0.05[mm]`	`Wire radius in coil`
`sigma_coil`	`5e7[S/m]`	`Wire conductivity`

2 Click **OK**.

3 From the **Options** menu, choose **Expressions>Global Expressions**.

4 In the **Global Expressions** dialog box, define the following variables with names and expressions. The description field is optional.

NAME	EXPRESSION	DESCRIPTION
`I_coil`	`1[A]`	`Current in coil`
`J_coil`	`I_coil*N/A`	`Equivalent current density in coil`

5 Click **OK**.

GEOMETRY MODELING

1 Choose **Draw>Specify Objects>Circle** to create a circle with the following properties:

NAME	RADIUS	BASE	R	Z
C1	0.03	Center	0	0

2 Choose **Draw>Specify Objects>Rectangle** to create rectangles with the following properties:

NAME	WIDTH	HEIGHT	BASE	R	Z
R1	5e-3	2e-2	Corner	0	-1e-2
R2	3e-3	2e-2	Corner	7.5e-3	-1e-2
R3	0.04	0.08	Corner	0	-0.04

3 Click the **Zoom Extents** toolbar button to see the objects.

4 Select the circle and the third rectangle (C1 and R3), and click the **Intersection** toolbar button.

5 Open the **Fillet/Chamfer** dialog box from the **Draw** menu.

6 In the dialog box, expand rectangle R1 and select Vertices 2 and 3. Then expand rectangle R2 and select all vertices there. Hold down the Ctrl key to do multiple selections.

7 In the **Radius** edit field type `5e-4`.

8 Click **OK**.

PHYSICS SETTINGS

Variables

1 From the **Options** menu, point to **Integration Coupling Variables** and click **Subdomain Variables**.

2 In the **Subdomain Integration Variables** dialog box, define two variables with integration order 4 and global destination according to the table below.

NAME	EXPRESSION IN SUBDOMAIN 3
V_coil	N*2*pi*r*I_coil/(A*sigma_coil*pi*r_coil^2)
A	1

3 Click **OK**.

Subdomain Settings

1 Open the **Subdomain Settings** from the **Physics** menu.

2 Select Subdomain 2 and click the **Load** button to open the **Materials/Coefficients Library** dialog box.

3 In the dialog box, select the **Soft Iron (without losses)** material under the `acdc_lib.txt` library. (You may have to scroll both vertically and horizontally to see the library name.)
4 Click **OK** to close the **Materials/Coefficients Library** dialog box.
5 Select Subdomain 3 and enter `J_coil` in the **External current density** edit field.
6 Click the **Init** tab, select all subdomains, and type `1` in the edit field for the initial value.
7 Click **OK**.

Boundary Conditions

1 Open the **Boundary Settings** from the **Physics** menu.
2 Select Boundaries 1, 2, and 4, then choose **Axial symmetry** from the **Boundary condition** list. For all other boundaries keep the default condition, **Magnetic insulation**.
3 Click **OK**.

MESH GENERATION

1 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.
2 Click the **Custom mesh size** option button and type `4` in the **Resolution of narrow regions** edit field.
3 Click **Remesh** and then **OK**.

COMPUTING THE SOLUTION

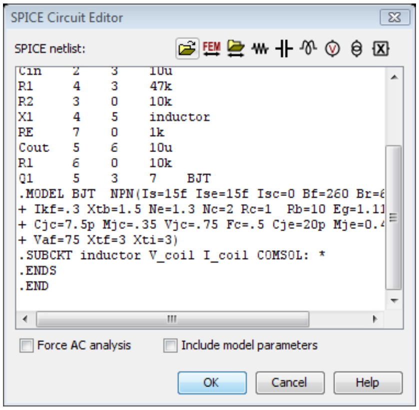
1 Click the **Solve** button.
2 From the **File** menu, choose **Save As**. Save the model to the file name `amplifier_and_inductor_nocircuit.mph`.

This completes the first part of the model.

SPICE IMPORT

1 From the **Physics** menu, choose **SPICE Circuit Editor**.
2 In the **SPICE Circuit Editor** dialog box, click on the **Load Netlist from File** toolbar button.
3 Browse to the file `amplifier.cir`, located in the model library path `AC/DC_Module/Electrical_Components/amplifier_and_inductor`.

4 Click **Import Netlist**. The entire file replaces the content of the **SPICE netlist** text area.



5 Click **OK** to close the **SPICE Circuit Editor** dialog box.

An alternative to importing text files is to manually type in the code in the SPICE circuit editor:

```
* BJT Amplifier circuit
.OPTIONS TNOM=27
.TEMP 27
Vin     1     0     sin(0 1 10kHz)
Vcc     4     0     15
Rg      1     2     100
Cin     2     3     10u
R1      4     3     47k
R2      3     0     10k
X1      4     5     inductor
RE      7     0     1k
Cout    5     6     10u
Rl      6     0     10k
Q1      5     3     7     BJT
.MODEL BJT  NPN(Is=15f Ise=15f Isc=0 Bf=260 Br=6.1
+ Ikf=.3 Xtb=1.5 Ne=1.3 Nc=2 Rc=1  Rb=10 Eg=1.11
+ Cjc=7.5p Mjc=.35 Vjc=.75 Fc=.5 Cje=20p Mje=0.4 Vje=0.75
+ Vaf=75 Xtf=3 Xti=3)
.SUBCKT inductor V_coil I_coil COMSOL: *
.ENDS
.END
```

The model is the same as before but with global variables and ODE variables added that represent the amplifier circuit. The syntax of these variables are related to the names in the file. The variable `I_R1_cir` stands for the current in resistor R1, `value_VCC_cir` is the voltage value of the supply voltage generator VCC, and

`U_X1_cir` is the voltage over the device X1, which is the inductor model you created earlier. The variables beginning with the letter `V` are node potentials referenced to the ground node, `0`, of the circuit.

As an alternative to the steps above, you can open the netlist from the model library path.

COMPUTING THE SOLUTION WITH A CIRCUIT

1 Open **Solver Parameters** from the **Solve** menu.
2 In the **Solver Parameters** dialog box, select **Parametric** from the **Solver** list.
3 In the **Parameter name** edit field type `value_VCC_cir`, and in the **Parameter values** edit field type `1:15`.
4 Click **OK**.
5 Click the **Restart** button.

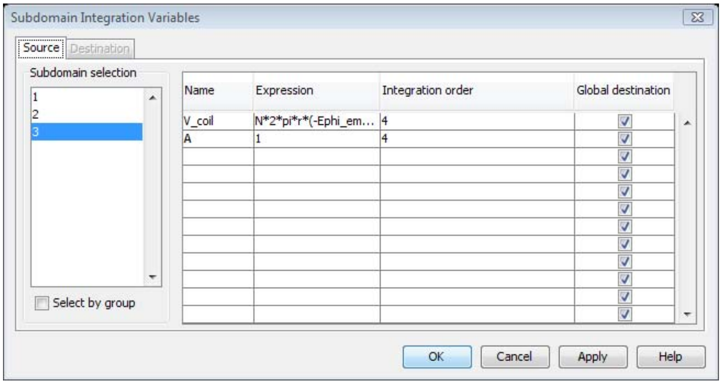
This solving step ramps the supply voltage stored in `value_VCC_cir` up to 15 V. The ramp is necessary to handle the highly nonlinear effects in the bipolar transistor model. The circuit is now at its bias point, and the plot should look like Figure 4-28 on page 192.

Transient Case

We will now make a transient calculation. In the previous static case we have no induced electric field, but when we change it to a transient case, the induced electric field needs to be added as a contribution to the coil voltage V_coil, in addition to the resistive effects. In the application mode the induced electric field is available as the variable $Ephi\_emqa$, and is computed as the time derivative of the magnetic potential.

1 From the **Options** menu point to **Integration Coupling Variables** and click **Subdomain Variables**.

2 In the **Subdomain Integration Variables** dialog box change the expression for the `V_coil` variable so it matches the expression below.



NAME	EXPRESSION IN SUBDOMAIN 3
`V_coil`	`N*2*pi*r*(I_coil/(sigma_coil*pi*r_coil^2)-Ephi_emqa)/A`

3 Click **OK**.

4 Open **Solver Parameters** from the **Solve** menu.

5 In the **Solver Parameters** dialog box select **Transient** from the **Analysis** list.

6 Type `linspace(0,5e-4,101)` in the **Times** edit field, type `1e-4` in the **Relative tolerance** edit field, and type `1e-6` in the **Absolute tolerance** edit field. The last steps are because the accuracy obtained with default error tolerances is not sufficient in this model.

7 Click **OK**.

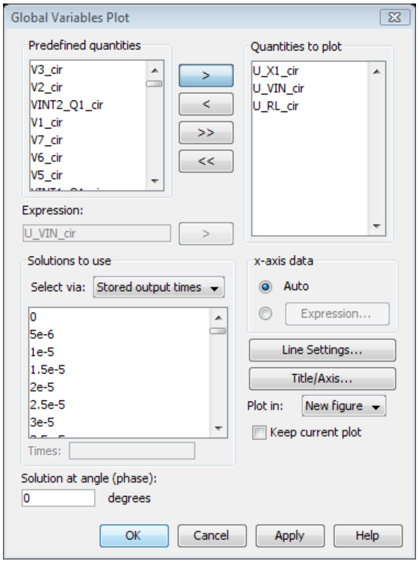
8 Click the **Restart** toolbar button.

POSTPROCESSING AND VISUALIZATION

1 Open **Global Variables Plot** from the **Postprocessing** menu.

2 From the **Predefined quantities** list select the variables **U_X1_cir**, **U_VIN_cir**, and **U_RL_cir**. Click the **>** button to add the selected variables to the **Quantities to plot** list.

3 From the **Solutions to use** list, select all the time steps from **4e-4** to **5e-4**.



4 Click the **Line settings** button. In the dialog box that appears, select **Cycle** from the **Line marker** list, and select the **Legend** check box. Click **OK**.

5 Click **OK** to see the plot in Figure 4-29 on page 193.

5

General Industrial Application Models

In this chapter you find models of general industrial applications that do not fit into any of the other chapters. Typically, these models deal with eddy currents, electric impedance, and other static or low-to-intermediate frequency applications.

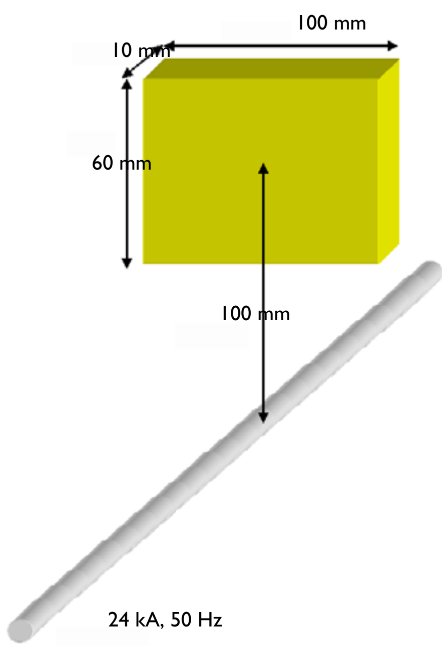
Eddy Currents in 3D

Introduction

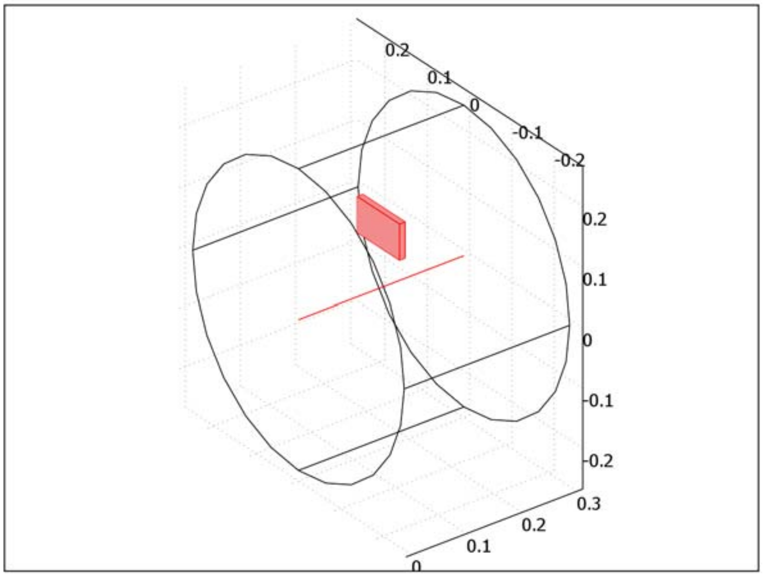
Induced eddy currents and associated thermal loads is of interest in many high power AC applications. This example is of general nature and illustrates some of the involved physics as well as suitable modeling techniques in the AC/DC Module.

Model Definition

A metallic plate is placed near a 50 Hz AC conductor. The resulting eddy current distribution in the plate depends on the conductivity and permeability of the plate. Four different materials will be considered: copper, aluminum, stainless steel, and magnetic iron.The model consists of a single wire and a plate with dimensions as shown below.



As you cannot afford meshing an infinite volume, it is necessary to specify a finite volume to mesh and solve for. In this case, it makes sense to enclose the wire and the plate in a cylinder with the wire on the axis of this cylinder.



The conductor is modeled as a line current with 0° phase and an effective value of 24 kA.

In the subdomains the magnetic vector potential is calculated from

$$(j\omega\sigma - \omega^2\varepsilon)\mathbf{A} + \nabla \times \left(\frac{1}{\mu}\nabla \times \mathbf{A}\right) = \mathbf{0}$$

where σ is the conductivity, ε the permittivity, and μ the permeability.

An important parameter in eddy current modeling is the skin depth, δ.

$$\delta = \sqrt{\frac{2}{\omega\mu\sigma}}$$

The table below lists the skin depth for the different materials at a frequency of 50 Hz.

MATERIAL	REL. PERMEABILITY	CONDUCTIVITY	SKIN DEPTH
Copper	1	$5.998 \cdot 10^7$ S/m	9 mm
Aluminum	1	$3.774 \cdot 10^7$ S/m	12 mm
Stainless steel	1	$1.137 \cdot 10^6$ S/m	67 mm
Iron	4000	$1.12 \cdot 10^7$ S/m	0.34 mm

The combination of a high permeability and a high conductivity can make it impossible to resolve the skin depth when modeling. When the skin depth is small in comparison to the size of conducting objects, the interior of those may be excluded from the model and replaced with an impedance boundary condition accounting for induced surface currents. This condition uses the following relation between the magnetic and electric field at the boundary:

$$\mathbf{n} \times \mathbf{H} + \sqrt{\frac{\varepsilon - j\sigma/\omega}{\mu}}\mathbf{n} \times (\mathbf{E} \times \mathbf{n}) = \mathbf{0}.$$

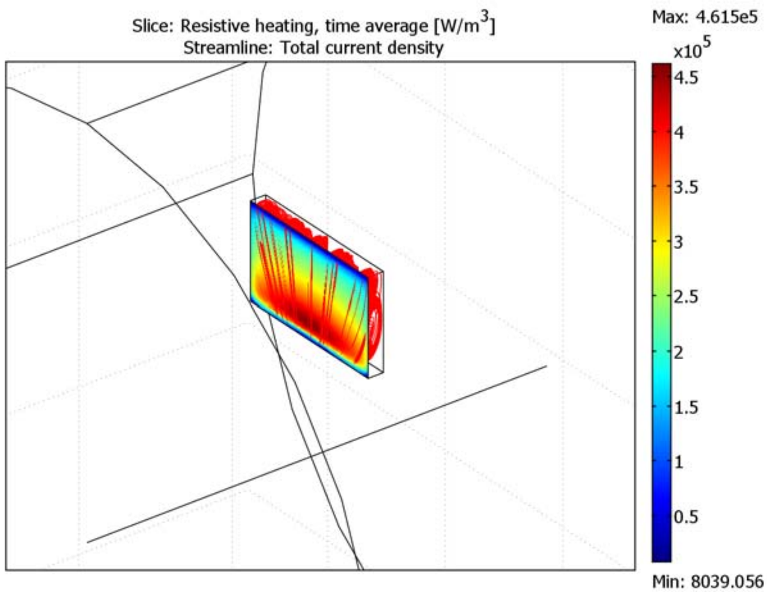
The distribution of the dissipated power, P_d, has the unit $\mathrm{W/m^2}$, and can be calculated from

$$P_d = \frac{1}{2}(\mathbf{J}_S \cdot \mathbf{E}^*)$$

where $\mathbf{J}_\mathrm{S}$ is the induced surface current density, and the asterisk denotes the complex conjugate.

Results and Discussion

The induced eddy current distribution for a plate made of copper is shown as streamlines, whereas the distribution of the ohmic losses is shown as a slice plot.



A total dissipated power of 6 W was obtained from integration through the plate. By repeating the simulation for different materials, the model shows that lowering the conductivity decreases the dissipated power. However, for high permeability materials like soft iron, the dissipated power is higher than in a copper despite a much lower conductivity.

Model Library path: `AC/DC_Module/General_Industrial_Applications/eddy_currents_3d`

MODEL NAVIGATOR

Create a new 3D model using the **AC/DC Module>Quasi-Statics, Magnetic> Induction Currents** application mode.

GEOMETRY MODELING

The entire geometry is drawn in mm for simplicity. The final step in the geometry creation is a conversion of all dimensions to meters because this model uses SI units. Use the toolbar buttons and the **Draw** menu.

2D Cross Section

The geometry is created from a 2D cross section, that lies in the *yz*-plane of the final geometry.

1 Begin by specifying a *yz* work plane at $x = 0$ from the **Draw** menu.
2 In the 2D work plane, draw a circle with a radius of 250 mm.
3 Add a point at the center of the circle.
4 Draw a 60 mm tall and 100 mm wide rectangle that is centered at $x = 0$, $y = 100$ mm.

Extruding to 3D

Create the 3D geometry from the cross section.

1 Select both the circle and the point and **Extrude** (from the **Draw** menu) these objects a distance of 300 mm.
2 Go back to the 2D view and **Extrude** the rectangle by 10 mm.
3 In the 3D view, select the resulting box and **Move** it by 145 mm in the *x* direction.

The next step is to convert the length scale to meters. Select all objects and **Scale** them with the factor `1e-3` in all directions.

PHYSICS SETTINGS

Open the **Application Scalar Variables** dialog from the **Physics** menu, and make sure that the **Frequency** is set to `50` Hz.

Subdomain Settings

Open the **Subdomain Settings** dialog box and select the plate. Click the **Electric Parameters** tab, click the **Load** button, and choose **Copper** from the **Basic Material Properties**. For the air domain, enter a small conductivity of `100` [S/m] to improve convergence for the iterative solver. Any parameters not specified can be left at their default values.

Boundary Conditions

Leave the boundary conditions at the default settings, which is magnetic insulation.

Edge Settings

Open the **Edge Settings** dialog box and specify a line current of `24e3*sqrt(2)` A for the conductor (Edge 6).

MESH GENERATION

In order to resolve the skin depth while maintaining a good mesh economy, proceed as follows:

1 Choose **Mesh>Free Mesh Parameters**.

2 On the **Subdomain** tab, select Subdomain 2 and set the **Maximum element size** to `5e-3`.

3 On the **Advanced** tab, set the **scale factor** to `0.4` in the **y** and **z** directions.

4 Click the **Mesh Selected** button.

5 On the **Global** tab click the **Reset to Defaults** button, then select **Predefined mesh sizes: Extremely coarse**.

6 On the **Subdomain** tab, select Subdomain 1 and set the **Maximum element size** to `0.1`.

7 Click the **Mesh Selected** button and then **OK** to close the dialog box.

These settings force a fine mesh in the plate but allow for a quick growth of the element size outside of the plate. In order not to get a too coarse mesh in the air, the settings limit the element size to 10 cm.

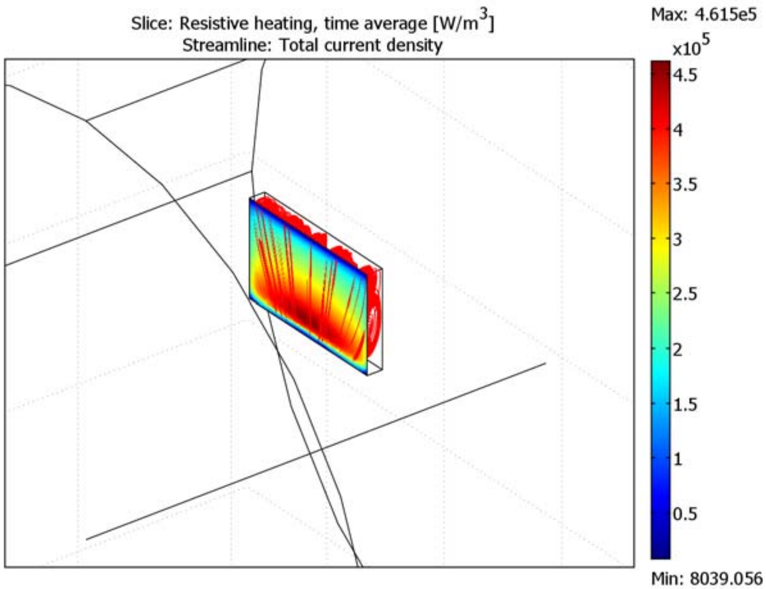
COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to solve the model.

POSTPROCESSING AND VISUALIZATION

The default plot is not what you want so open the **Plot Parameters** dialog box. Create a slice plot of the **Resistive heating, time average (Qav_emqa)** at x `= 0.1451` only, and do a streamline plot of the **Total current density**. Change the **Number of start points** to

50. In order to plot the streamlines only in the plate, **Suppress** the air domain from the **Options** menu.



The final step is to calculate the total power dissipated in the plate by the induced eddy currents. This is done by going to the **Postprocessing** menu and select **Subdomain Integration**. In the dialog that appears, select the plate subdomain and from the list of **Predefined quantities** select **Resistive heating, time average (Qav_emqa)** before clicking **OK**. The result should be close to 6 W.

Changing to Aluminum and Stainless Steel

Repeating the analysis for aluminum and stainless steel is simple—just change the material properties for the plate subdomain and solve again. Aluminum is available from the Basic Material Properties whereas you have to enter stainless steel by typing the corresponding conductivity `1.137e6` (S/m) value in the dialog box.

Changing to Magnetic Iron

For magnetic iron, the skin depth is too small to be resolved with reasonable computational effort. The solution is to use the impedance boundary condition.

Subdomain Settings

1 Open the **Subdomain Settings** dialog box from the **Physics** menu.

2 Select the plate, then clear the **Active in this domain** check box. This tells the solver not to include this domain in the solution process and also makes the impedance boundary condition available on the boundaries of this domain.

3 Click **OK**.

Boundary Conditions

1 Open the **Boundary Settings** dialog box from the **Physics** menu.

2 Select the plate's boundaries, then select **Impedance boundary condition** from the **Boundary condition** list.

3 Click the **Material Properties** tab, then click the **Load** button and select **Iron** in the **Basic Materials Properties** list in the **Materials/Coefficients Library** dialog box that appears. Click **OK**.

4 Click **OK**.

COMPUTING THE SOLUTION

Use the same settings as before. Click the **Solve** button on the Main toolbar to solve the model. This takes a few minutes.

POSTPROCESSING AND VISUALIZATION

The default plot may now give an error message as the plotted quantity no longer is defined in the plate domain.

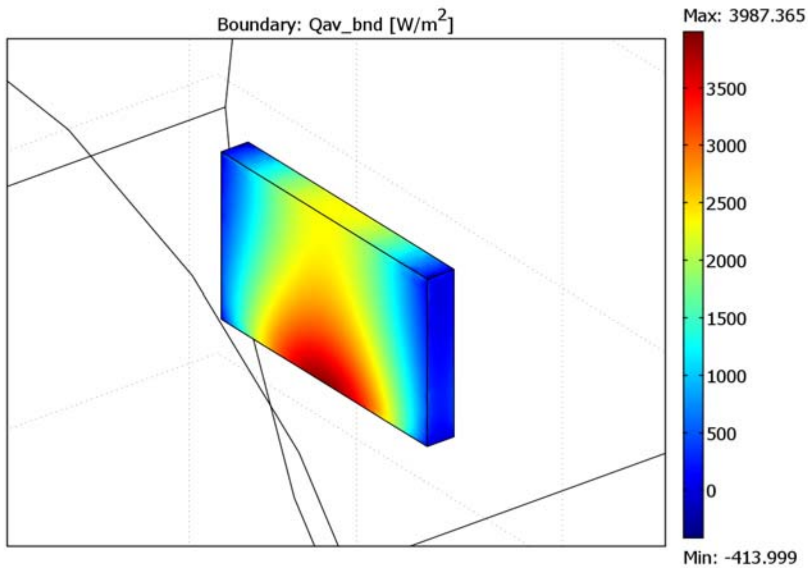
1 Click the **Plot Parameters** button on the Main toolbar.

2 On the **General** page, clear the **Slice** and **Streamline** check boxes and select the **Boundary** check box in the **Plot type** area.

3 Click the **Boundary** tab. In the **Boundary data** area, select **Surface resistive heating, time average** from the **Predefined quantities** list to generate a plot of the resistive heating.

4 To see the results on the plate, point to **Suppress** in the **Options** menu.

5 Select boundaries **1-5, 12** and click **OK**.



The final step is to calculate the total power dissipated in the plate.

1 From the **Postprocessing** menu, select **Boundary Integration**.

2 In the **Expression to integrate** area, select **Surface resistive heating, time average** from the **Predefined quantities** list.

3 Click **OK**.

The result displayed in the message log should be close to 27 W.

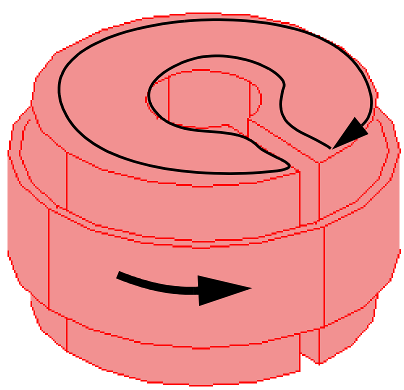
Cold Crucible[1]

Introduction

A cold crucible is used for the manufacturing of alloys that require a high degree of purity, like titanium alloys for the aeronautic industry. The cold crucible in this model is made of several water-cooled copper sectors forming the container in which the alloy is manufactured.

Model Definition

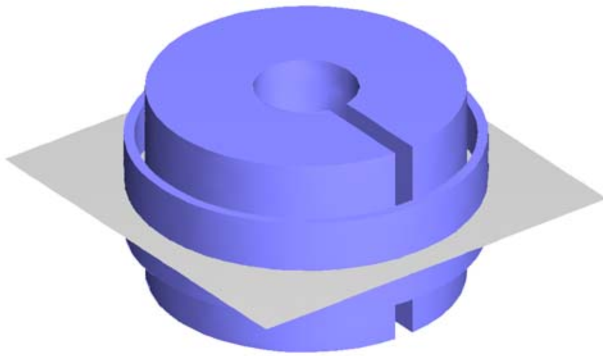
The crucible is sketched in the figure below. The harmonic current flowing in the coils induces a current flowing in the opposite direction along the edge of the crucible. The induced currents produced in each of the sectors make the cold crucible act as field concentrator.



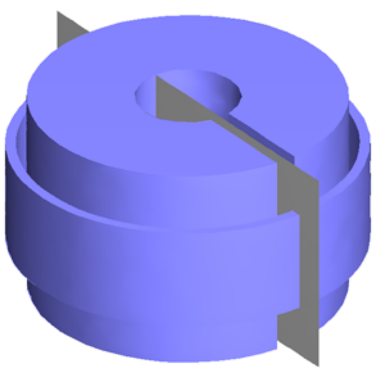
To save computation time, exploit the symmetries of the model and set up only one quarter of the crucible. The flow lines of the magnetic field are normal to the symmetry plane inserted in the figure below. To understand this, note that the geometry is symmetric with respect to the plane, and that the current is tangential to the plane.

1. Model made in cooperation with Roland Ernst, Centre National de la Recherche Scientifique, Grenoble, France.

The first symmetry can be achieved by letting this plane be a boundary of the model. To this boundary, apply a boundary condition setting the tangential component of the magnetic field to zero.



The following figure indicates a second symmetry plane. As the flow lines of the magnetic field make closed loops around the coils, the field must be tangential to the symmetry plane.



To model this symmetry, use a boundary condition that sets the tangential component of the magnetic potential to zero.

The crucible operates at 10 kHz. At this frequency the currents flow very close to the surface of the conducting regions. The skin depth

$$\delta = \sqrt{\frac{2}{\omega\mu\sigma}}$$

is less than one millimeter, and the inner radius of the crucible is 5 cm. You can therefore model the currents as surface currents.

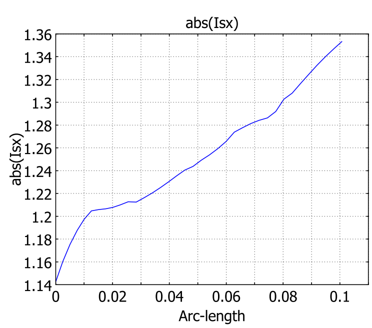
Because the electric potential does not need to be specified to generate currents, it is possible to work with the potential

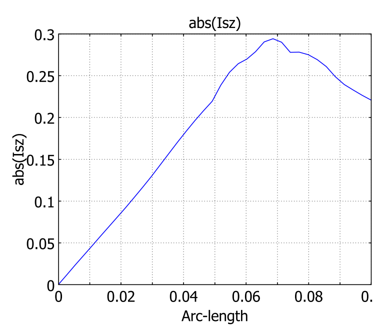
$$\tilde{\mathbf{A}} = \mathbf{A} - \frac{j}{\omega}\nabla V$$

and thereby reduce the set of four equations for the **A** and *V* potentials to three equations for the $\tilde{\mathbf{A}}$ potential (see "3D and 2D Quasi-Statics Application Modes" on page 154 in the *AC/DC Module User's Guide*).

Results and Discussion

To compute the surface currents flowing into and out of the end of the crucible, you can use integrals at that boundary. From Kirchhoff's current law the total current flowing into the boundary should be equal to the currents flowing out of it. The figure below shows the integrations along the *x* side and *z* side, where the start and end points of each curve correspond to the flow in and out of the boundary, respectively.





The total inflow to the boundary is 1.38 A and the total outflow is 1.40 A, which agrees with Kirchhoff's power law within 2%.

Model Library path: `ACDC_Module/General_Industrial_Applications/crucible`

MODEL NAVIGATOR

1 In the **Model Navigator**, select **3D** from the **Space dimension** list.

2 Then click on **AC/DC Module>Quasi-Statics, Magnetic>Induction Currents** to select the Quasi-Statics application mode.

3 Click **OK**.

OPTIONS AND SETTINGS

1 In the **Constants** dialog box in the **Options** menu, enter the following names and expressions.

NAME	EXPRESSION	DESCRIPTION
`sigMetal`	`5e7[S/m]`	`Conductivity of metal`
`sigAir`	`1[S/m]`	`Conductivity of air`
`Js0`	`16.5[A/m]`	`Coil surface current`

The electric conductivity $\sigma = 1$ S/m for air is obviously not the physically correct value, but too large a contrast between the conductivities makes it difficult for the numerical algorithms to find a solution. The error you make by using an incorrect value for the conductivity in air is negligible for the currents on the surfaces.

2 In the **Application Scalar Variables** dialog box in the **Physics** menu, set the frequency variable `nu_emqav` to `10000`.

GEOMETRY MODELING

Draw the crucible and inductor in a 2D work plane and then extrude them into 3D.

1 Create a work plane by opening the **Work-Plane Settings** dialog box in the **Draw** menu and clicking **OK** to obtain the default work plane in the *xy*-plane.

2 Open the **Axes/Grid Settings** dialog box from the **Options** menu. Enter the axis settings in the table below on the **Axis** tab.

3 Then click on the **Grid** tab and clear the **Auto** check box and enter the grid settings according to the table below.

AXIS		GRID	
x min	`-0.3`	x spacing	`0.1`

x max	`0.3`	Extra x	`-0.17`
			`-0.16`
			`0.155`

AXIS		GRID	
y min	`-0.2`	y spacing	`0.05`
y max	`0.2`	Extra y	`0.01`

4 Draw a circle C1 with radius 0.17 and a circle C2 with radius 0.16, both centered at the origin.

5 Select C1 and C2 and click the **Difference** button to create the solid object CO1.

6 Draw a circle C1 with radius 0.15 and a circle C2 with radius 0.05, both centered at the origin.

7 Select C1 and C2 and click the **Difference** button to create the solid object CO2.

8 Draw a rectangle R1 with the lower left corner at (–0.2, –0.2), **Width** `0.4`, and **Height** `0.2`.

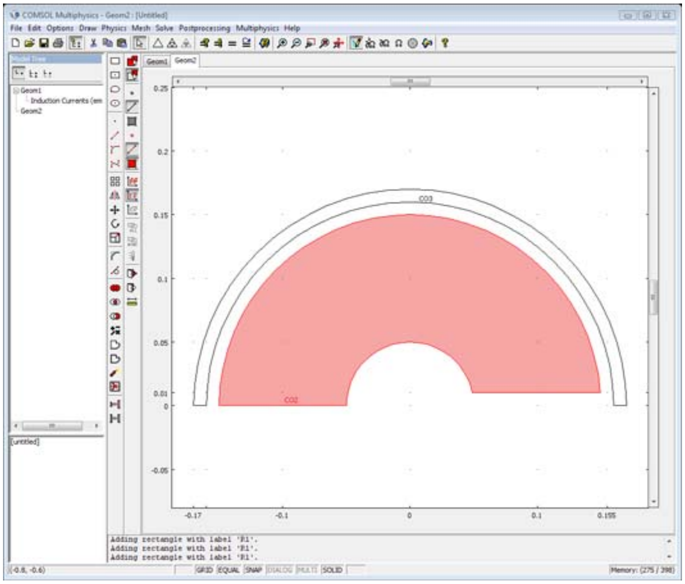
9 Open the **Create Composite Object** dialog box and enter the set formula `CO1-R1` to create the composite object CO3.

10 Draw a new rectangle R1 with the lower left corner at (–0.2, –0.2), **Width** `0.4`, and **Height** `0.2`.

11 Open the **Create Composite Object** dialog box and enter the set formula `CO2-R1` to create the composite object CO1.

12 Draw a rectangle R1 with the lower left corner at (0, 0), **Width** `0.155`, and **Height** `0.01`.

13 Open the **Create Composite Object** dialog box and enter the set formula `CO1-R1` to create the composite object CO2.



14 Select **Extrude** from the **Draw** menu to open the **Extrude** dialog box. Select the object CO3 and set the **Distance** to 0.05. Click **OK** to close the dialog box and create the extruded object EXT1.

15 Click on the **Geom2** tab to return to the work plane.

16 Open the **Extrude** dialog box again, and select the object CO2 and set the **Distance** to 0.1 to create the extruded object EXT2.

17 Now create a region of air surrounding the crucible. Press the **Block** button to open the **Block** dialog box and set the **Length** coordinates to (1, 0.5, 0.5). Click **More** and set the coordinates of the **Axis base point** to (-0.5, 0, 0). Click **OK** to close the dialog box and create the block BLK1.

18 Press the **Cylinder** button to open the **Cylinder** dialog box and set the **Radius** to 0.5 and the **Height** to 0.5. Click **OK** to close the dialog box and create the cylinder CYL1.

19 Open the **Create Composite Object** dialog box and enter the set formula `BLK1*CYL1`. Click the **Apply** button to create the object CO1.

20 Enter the set formula `CO1-EXT1-EXT2` and click **OK**.

PHYSICS SETTINGS

Boundary Conditions

Enter boundary conditions according to the following table. Note that the impedance boundary condition has a **Material Properties** tab where you enter the conductivity. For the surface current, enter the x, y, and z components, which are $J_{s0}(-\sin\alpha, \cos\alpha, 0)$, where α is the azimuthal angle.

SETTINGS	BOUNDARIES 1, 2, 4, 16	BOUNDARIES 3, 8	BOUNDARIES 5, 6, 7, 14, 15	BOUNDARIES 9–13, 17
Boundary condition	Magnetic insulation	Electric insulation	Surface current	Impedance boundary condition
J_s, x comp.			`(-y/sqrt(x^2+y^2))*Js0`	
J_s, y comp.			`(x/sqrt(x^2+y^2))*Js0`	
J_s, z comp.			0	
σ				`sigMetal`
μ_r				1
ε_r				1

Subdomain Settings

On the **Electric Parameters** tab, enter properties according to the following table.

SETTINGS	SUBDOMAIN 1
σ	`sigAir`

OPTIONS AND SETTINGS

After solving the problem, the postprocessing includes a study of how the current flows on the rectangular end of the crucible (Boundary 17). To prepare for that, define two coupling variables constituting the integrated current over a cross section of this boundary. `Jsx_emqav` is the x component of the surface current, and `Isx` is the integral of `Jsx_emqav` in the vertical direction across Boundary 17,

$$I_{sx}(x) = \int J_{sx}(x,z)dz$$

`Jsz_emqav` is the z component of the surface current, and `Isz` is the integral of `Jsz_emqav` in the horizontal direction across Boundary 17,

$$I_{sz}(z) = \int J_{sz}(x,z)dx$$

The table below summarizes the variable definitions.

VARIABLE	TYPE	SOURCE	DESTINATION
`Isx`	Projection	Boundary: 17 Integrand: `Jsx_emqa` Integration order: 2 Source transformation, x: x Source transformation, y: z	Edge: 38 Destination transformation: x
`Isz`	Projection	Boundary: 17 Integrand: `Jsz_emqa` Integration order: 2 Source transformation, x: z Source transformation, y: x	Edge: 40 Destination transformation: z

1 In the **Boundary Projection Variables** dialog box, select Boundary 17, enter the **Name** `Isx`, set the **Expression** to `Jsx_emqa`, and use the **Integration order** `2`. Select the **General transformation** and fill in the **x** and **y** edit fields with `x` and `z`, respectively. The integration will be performed over the second coordinate, z, and `Isx` will therefore be a function of x.

2 On the **Destination** page, set the **Level** to **Edge**, and select Edge 38, which is one of the edges of Boundary 17, and set the **Destination transformation** to `x`.

3 Go back to the **Source** page and add a second projection variable, `Isz`, select Boundary 17 and set the **Integrand** to `Jsz_emqa`, the **Integration order** to `2`. Select the **General transformation**, and fill in `z` and `x` in the fields **x** and **y**. In this case the integration will be performed over x.

4 On the **Destination** page, select Edge 40, and set the **Destination transformation** to `z`.

MESH GENERATION

To get a good quality mesh, set the maximum mesh element size manually on the boundaries adjacent to the thin regions in the model.

1 Open the **Free Mesh Parameters** dialog box

2 Go to the **Boundary** page, select Boundaries 5, 7, 9, 13, 14, and 15 and set the **Maximum element size** to `0.04`.

3 Initialize the mesh.

COMPUTING THE SOLUTION

1 Press the **Solve** button. (use the default solver)

POSTPROCESSING AND VISUALIZATION

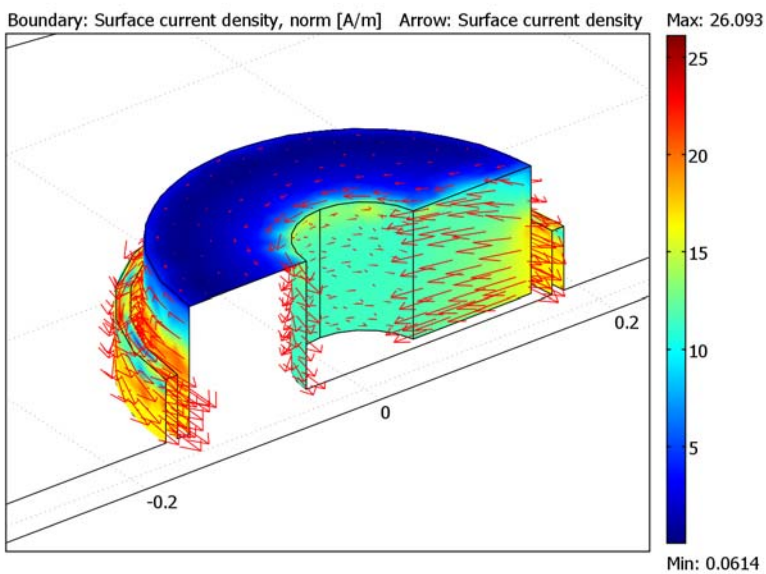
1 To see the currents on the crucible, open the **Suppress Boundaries** dialog box from the **Options** menu and suppress Boundaries 1–4, 8, and 16.

2 Open the **Plot Parameters** dialog box.

3 On the General page, clear the **Slice** check box and select the **Boundary** and **Arrow** check boxes in the **Plot type** area.

4 Click the **Boundary** tab and select **Surface current density, norm** from the **Predefined quantities** list on the **Boundary Data** page.

5 Click the **Arrow** tab. From the **Plot arrows on** list, select **Boundaries**. In the **Arrow parameters** area, set the **Arrow type** to **arrow**. Clear the **Auto** check box, and set the **Scale factor** to 3.



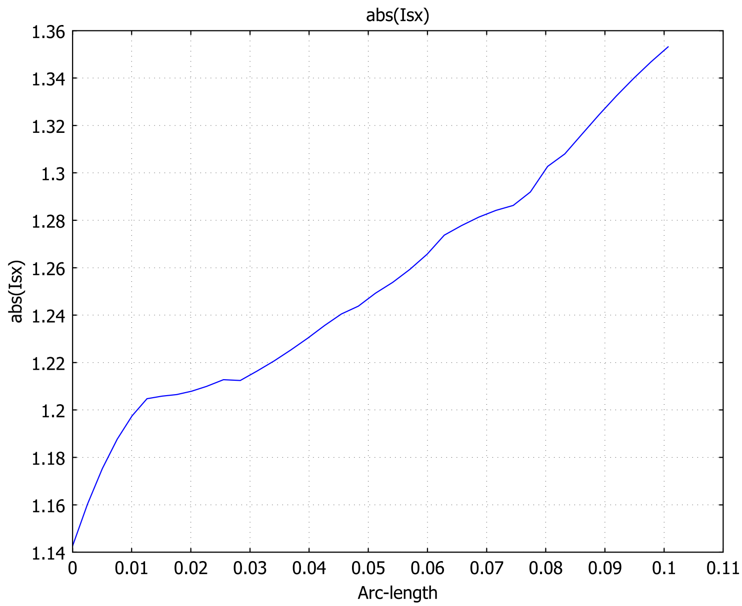
To verify the current conservation on the end boundary of the crucible, Boundary 17, make use of the coupling variables `Isx` and `Isz`:

1 From the **Postprocessing** menu, open the **Domain Plot Parameters** dialog box.

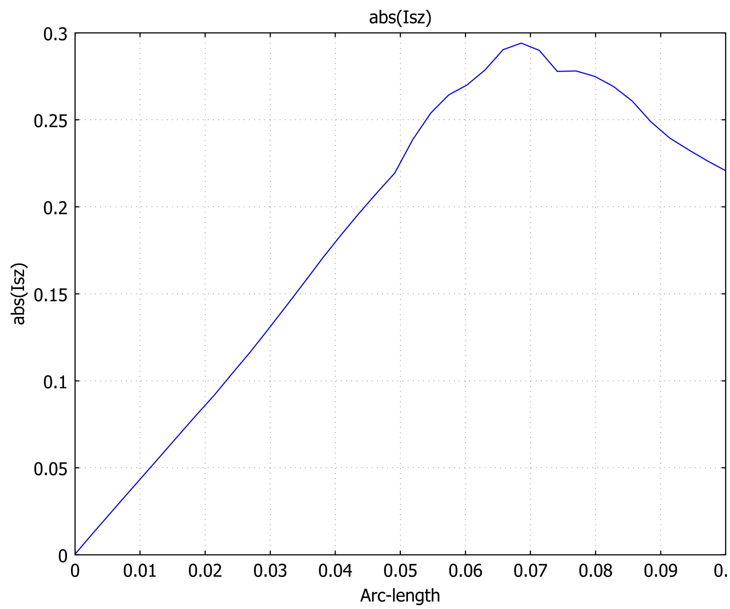
2 Click the **Line** tab and select Edge 38. Enter the **Line expression** `abs(Isx)`, then click **Apply** to generate the plot.

3 The left side of the plot corresponds to $x = 0.15$ and the right side to $x = 0.05$. The value at $x = 0.15$ is the total current flowing in through the right edge of the

boundary, and the value at $x = 0.05$ is the total current flowing out through the left edge of the boundary.



4 Select Edge 40 and change the **Line expression** to `abs(Isz)`. The left side of the plot corresponds to z = 0 and the right side to z = 0.1. The value at z = 0.1 is the total current flowing out through the top edge of the boundary.



5 By studying the plots, you can see that the current flowing in equals the current flowing out. To more accurately verify this, export the FEM structure to the COMSOL Script or MATLAB workspace by selecting **Export FEM structure as 'fem'** from the **File** menu. (This requires that you run COMSOL Multiphysics together with COMSOL Script or MATLAB.) Then enter the commands

```
Iinx=postinterp(fem,'abs(Isx)',0.1,'dom',38);
Iinz=postinterp(fem,'abs(Isz)',0.0,'dom',40);
Ioutx=postinterp(fem,'abs(Isx)',0.0,'dom',38);
Ioutz=postinterp(fem,'abs(Isz)',0.1,'dom',40);
(Ioutx+Ioutz)/(Iinx+Iinz)

ans =

    1.0322
```

The evaluation points along the edges are given in the local coordinate of the edge. As the length of these edges is 0.1, the coordinate lie in the range 0 to 0.1. The current flowing in and the current flowing out differ by less than two percent.

Electric Impedance Sensor

Introduction

Electric impedance measurements are used for imaging and detection. The applications range from nondestructive testing and geophysical imaging to medical imaging. One example is to monitor the lung function of babies in neonatal intensive care. The frequency ranges from less than 1 Hz to about 1 GHz depending on the application.

This model simulates a single electrode placed on a conducting block with an air filled cavity inside. The block is connected to ground on the lower face and on the sides. The analysis shows how the lateral position of the cavity affects the measured impedance, which you compute in a postprocessing step.

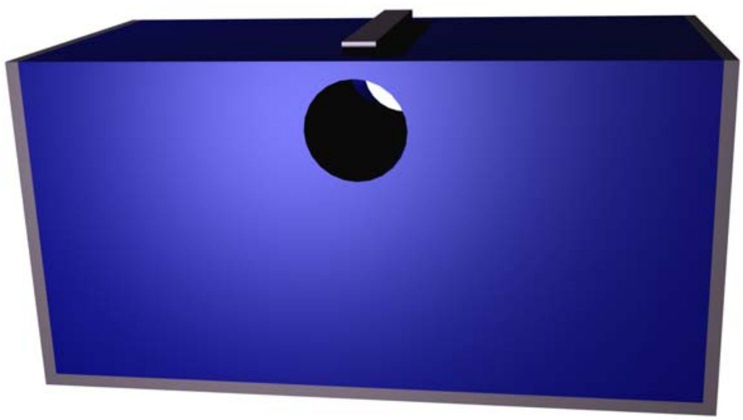


Figure 5-1: The electrode is placed on a conducting block with an air-filled cylindrical cavity inside.

Model Definition

In the AC/DC Module you solve the problem using the 2D Small In-Plane Currents application mode. This application mode is useful for the modeling of AC problems when inductive effects are negligible. A sufficient requirement for this is that the skin depth is large compared to the size of the geometry. The skin depth, δ, is given by

$$\delta = \sqrt{\frac{2}{\omega\mu\sigma}}$$

where ω is the angular frequency, μ is the permeability, and σ is the conductivity. This model uses nonmagnetic materials with a frequency of 1 MHz and a typical electric conductivity of 1 mS/m, so the skin depth is about 15 m. The size of the geometry is about 1 m.

DOMAIN EQUATIONS

When induction is neglected, the electric field is curl free and can be assigned a scalar potential, V. The equation of continuity for the conduction and displacement currents then becomes

$$-\nabla \cdot ((\sigma + j\omega\varepsilon_r\varepsilon_0)\nabla V) = 0$$

where ε_0 is the permittivity of free space, and ε_r is the relative permittivity. The electric field $\mathbf{E}$ and displacement $\mathbf{D}$ are obtained from the gradient of V:

$$\mathbf{E} = -\nabla V$$
$$\mathbf{D} = \varepsilon_0\varepsilon_r\mathbf{E}$$

BOUNDARY CONDITIONS

Ground potential boundary conditions are applied on the lower and vertical edges of the domain. The upper edge is set to insulation except at the electrode, where a uniformly distributed current source of 1 A is applied.

$$\mathbf{n} \cdot \mathbf{J} = J_n$$

Results and Discussion

Figure 5-2 shows the calculated impedance and its phase angle as functions of air cavity coordinate. When the cavity passes under the electrode, a sharp peak appears in the impedance value.

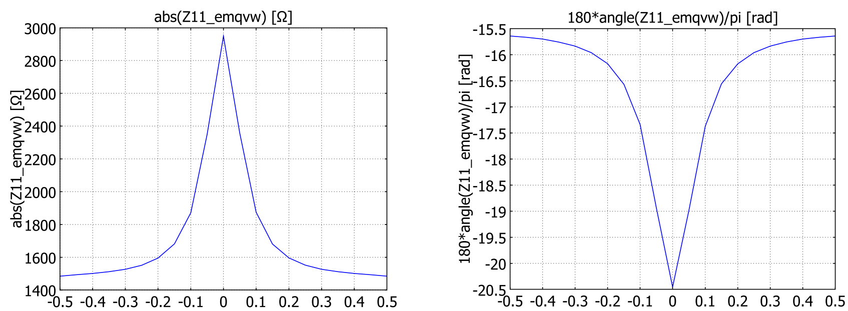


Figure 5-2: Absolute value and phase for the impedance as functions of air cavity coordinate. The peaks occur when the cavity is beneath the electrode.

Model Library path: `ACDC_Module/General_Industrial_Applications/electric_impedance`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Select **2D** in the **Space dimension** list.
2. In the **AC/DC Module>Quasi-Statics, Electric>In-Plane Electric Currents** application mode.
3. Click **OK**.

OPTIONS AND SETTINGS

1. Select **Axes/Grid Settings** from the **Options** menu to open the **Axes/Grid Settings** dialog box.
2. On the **Axis** tab set **x min** to `-0.55` and **x max** to `0.55`. Set the interval limits on the y-axis to `-0.55` and `0`.

3 Click the **Grid** tab and then clear the **Auto** check box to manually define new grid settings.

4 Set **x spacing** to `0.1` and give the values `-0.01 0.01` in the **Extra x** text field. Set **y spacing** to `0.1`.

5 To define some global constants to be used in your model, select **Constants** from the **Options** menu.

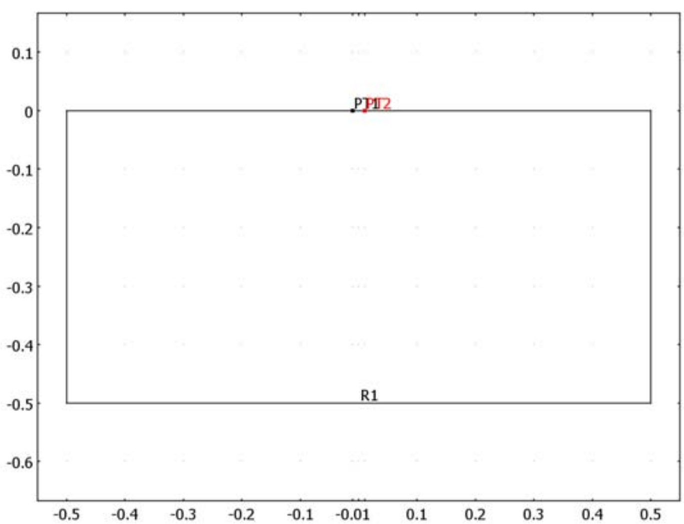
6 In the **Constants** dialog box, enter the following variable names, expressions, and (optionally) descriptions:

NAME	EXPRESSION	DESCRIPTION
`sig_bulk`	`1[mS/m]`	`Bulk conductivity`
`eps_r_bulk`	`5`	`Relative permittivity in bulk`
`x0`	`0[m]`	`x position of cavity center`
`y0`	`-0.1[m]`	`y position of cavity center`
`r0`	`0.09[m]`	`Cavity radius`

GEOMETRY MODELING

1 Click the **Rectangle/Square** button on the Draw toolbar or select the corresponding entry in the **Draw** menu, then draw a rectangle with opposite corners at (–0.5, –0.5) and (0.5, 0), using the left mouse button.

2 Click the **Point** button on the Draw toolbar and draw a point at (–0.01, 0) and another point at (0.01, 0).



PHYSICS SETTINGS

Scalar Variables

Open the dialog box by selecting **Scalar Variables** from the **Physics** menu and set the frequency **nu_emqvw** to `1e6`.

Boundary Conditions

Ground potential boundary conditions are applied on the lower and vertical edges of the domain. The upper edge is set to insulation except at the electrode, where a port is defined using the fixed current density property. This applies a uniformly distributed current source of 1 A to the electrode. The following table shows the corresponding boundary settings:

SETTINGS	BOUNDARY 4	BOUNDARIES 1, 2, 6	BOUNDARIES 3, 5
Boundary condition	Port	Ground	Electric insulation
Port number	1		
Use port as input	selected		
Input property	Fixed current density		

Subdomain Settings

To model the cavity, use functions and logical expressions for the permittivity and conductivity distributions. This has the advantage that you can change the position of the cavity without modifying the geometry. Use the default constitutive relation,

$$\mathbf{D} = \varepsilon_0\varepsilon_r\mathbf{E}$$

and enter the material parameters according to the table below.

SETTINGS	SUBDOMAIN 1
σ	`sig_bulk*(((x-x0)^2+(y-y0)^2)>r0^2)`
ε_r	`1+(eps_r_bulk-1)*(((x-x0)^2+(y-y0)^2)>r0^2)`

MESH GENERATION

1. Open the **Free Mesh Parameters** dialog box.
2. Select **Normal** from the **Predefined mesh sizes** list on the **Global** page. Select **Custom mesh size** and specify a **Maximum element size** of `0.01`.
3. Initialize the mesh.

COMPUTING THE SOLUTION

1 In the **Solver Parameters** dialog box, select **Parametric** in the **Solver** list.

2 Type `x0` in the **Parameter name** edit field and `linspace(-0.5,0.5,21)` in the **Parameter values** edit field. Click **OK**.

3 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

After solving the problem, a surface plot for the electric potential appears. You can use the buttons on the Plot toolbar to get other types of plots. To visualize the current distribution on a dB scale, open the **Plot Parameters** dialog box.

1 Select **Plot Parameters** from the **Postprocessing** menu to open the **Plot Parameters** dialog box.

2 Select **Surface** as **Plot type** and set **Parameter value** to `0` on the **General** page.

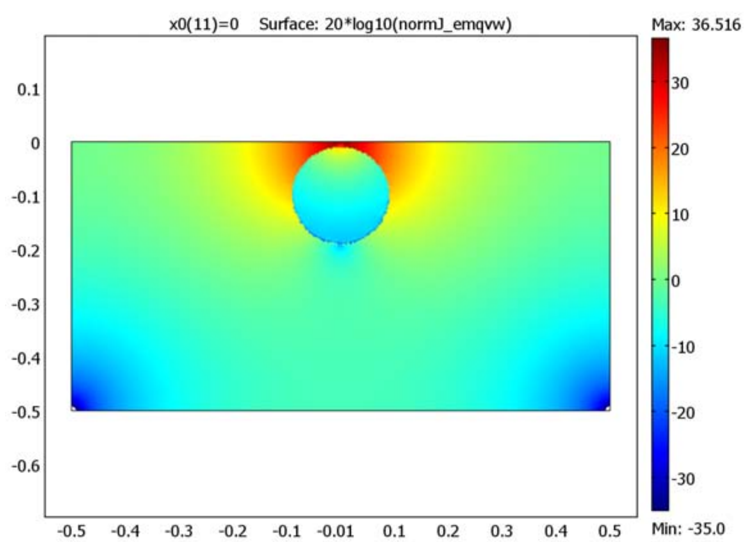
3 On the **Surface** page, set the **Surface Expression** to `20*log10(normJ_emqvw)`.

4 Click **Apply** to generate a plot.

5 To better see the variations the color range can be changed. Click **Range** to open the **Color Range** dialog box.

6 Clear the **Auto** check box to manually define new range settings.

7 Keep the value for **Max** but change the **Min** value to `-35`. Click **OK**.

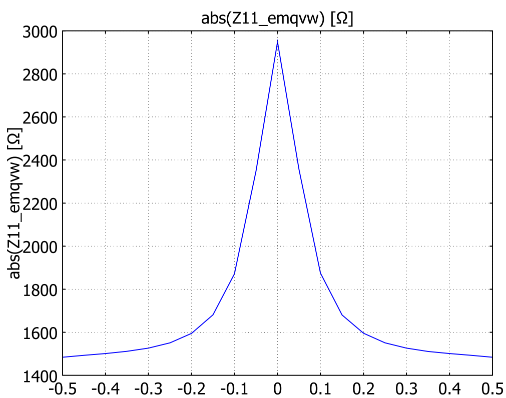


The impedance is defined as the ratio of voltage to total current at the electrode. You can visualize it as a function of the cavity position using the cross-sectional domain plot tool available from the **Postprocessing** menu. The impedance is available as a postprocessing variable, `Z11_emqvw`.

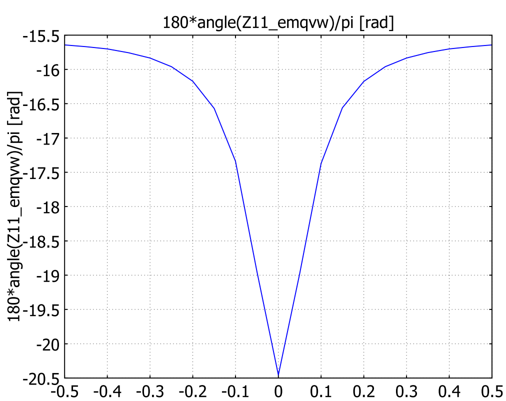
1 Open the **Domain Plot Parameters** dialog.

2 Select all solutions from the **Solution to use** list by pressing Ctrl+A.

3 Now select the **Point** tab and set **Expression** to `abs(Z11_emqvw)` and select Point 1. Click **Apply** to get the following figure.



4 Finally, repeat the last step but set the **Expression** to `180*angle(Z11_emqvw)/pi` to get the phase angle of the impedance.



One-Sided Magnet and Plate

Introduction

Permanent magnets with a one-sided flux are used to attach posters and notes to refrigerators and notice boards but can also be found in advanced physics applications like particle accelerators. The one-sided flux behavior is obtained by giving the magnet a magnetization that varies in the lateral direction (Ref. 1). As no currents are present, it is possible to model a permanent magnet using a scalar magnetic potential formulation. This model shows this technique to model a cylindrical one-sided permanent magnet. A special technique to model thin sheets of high permeability material was used to model a thin µ-metal plate next to the magnet. This circumvents the difficulty of volumetric meshing of thin extended structures in 3D.

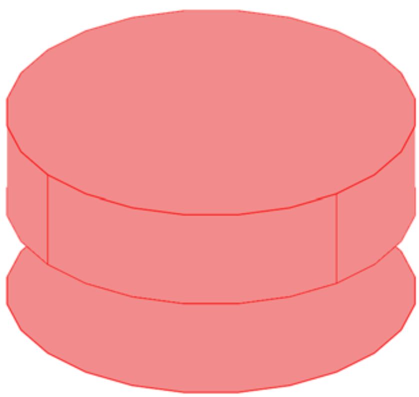


Figure 5-3: A cylindrical magnet above a µ-metal plate is modeled.

Model Definition

In a current free region, where

$$\nabla \times \mathbf{H} = \mathbf{0}$$

you can define the scalar magnetic potential, V_m, from the relation

$$\mathbf{H} = -\nabla V_m$$

This is analogous to the definition of the electric potential for static electric fields.

Using the constitutive relation between the magnetic flux density and magnetic field

$$\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$$

together with the equation

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

you can derive the following equation for V_m:

$$-\nabla \cdot (\mu_0 \nabla V_m - \mu_0 \mathbf{M}) = 0$$

It can be shown that applying a laterally periodic magnetization of

$$\mathbf{M} = (M_{\mathrm{pre}} \sin(kx), 0, M_{\mathrm{pre}} \cos(kx))$$

results in a magnetic flux that only emerges on one side of the magnet.

Results and Discussion

Figure 5-4 shows the calculated magnetic flux density and direction for the one-sided magnet. The resulting force on the plate is considerably higher for the case with the one-sided magnetization compared to the case with uniform magnetization of the same amplitude.

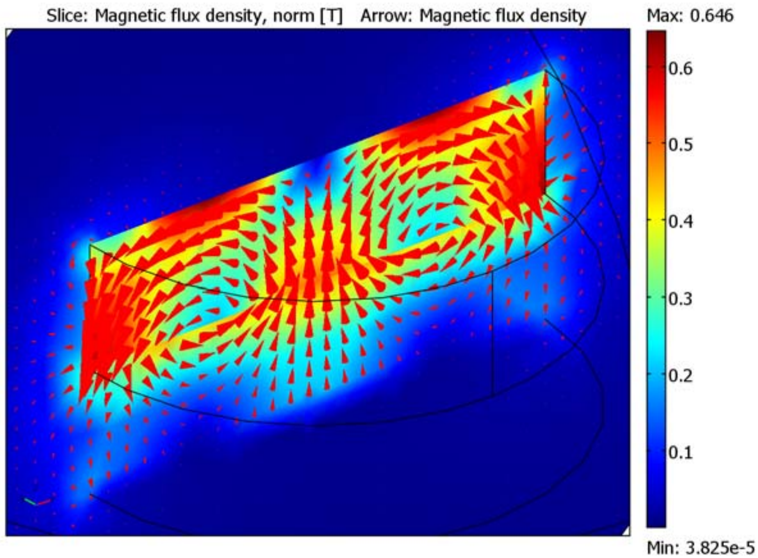


Figure 5-4: The magnetic flux density and direction is plotted in a cross section of the geometry. The one-sided behavior is clearly seen as the flux does not emerge on the top of the magnet.

Reference

1. Shute, Mallinson, Wilton and Mapps, "One-Sided Fluxes in Planar, Cylindrical and Spherical Magnetized Structures," *IEEE Transactions on Magnetics*, Vol 36, No. 2, March 2000, pp. 440–451.

Model Library path: `ACDC_Module/General_Industrial_Applications/one_sided_magnet`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **3D** from the **Space dimension** list.

2 Select the **AC/DC Module>Statics>Magnetostatics, No Currents** application mode.

3 Click **OK**.

OPTIONS AND SETTINGS

In the **Constants** dialog box enter the following names and expressions. The description field is optional and can be omitted.

NAME	EXPRESSION	DESCRIPTION
`k`	`pi/10[mm]`	`Wave number in x direction`
`M_pre`	`5e5[A/m]`	`Magnetization amplitude in magnet`
`mu_r`	`4e4`	`Relative permeability in plate`
`th`	`0.5[mm]`	`Plate thickness`

GEOMETRY MODELING

1 Use the **Cylinder** tool to create a cylinder with the radius 0.01 and the height 0.005.

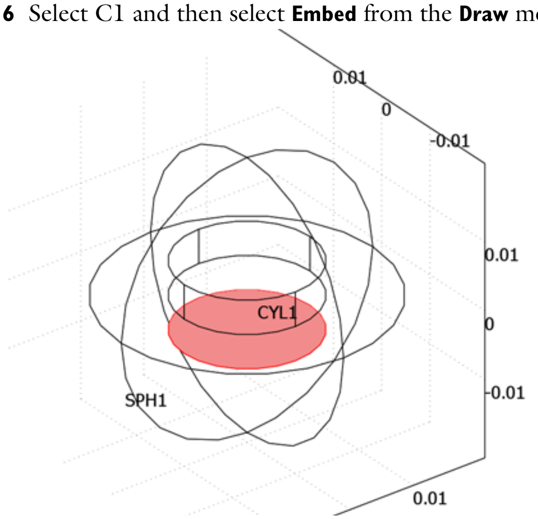
2 Use the **Sphere** tool to create a sphere with the radius 0.02.

3 Open the **Work-Plane Settings** dialog from the **Draw** menu and use the default work plane in the *xy*-plane. Set **z** to -0.005 and click **OK**.

4 First click the **Projection of All 3D Geometries** toolbar button and then click the **Zoom Extents** toolbar button to adapt the axis settings of the drawing board.

5 Draw a circle C1 with radius 0.01 centered at (0, 0).

6 Select C1 and then select **Embed** from the **Draw** menu and click **OK**.



PHYSICS SETTINGS

Subdomain Settings

In the **Subdomain Settings** dialog box, change the **Constitutive relation** in Subdomain 2 to allow for a premagnetization vector **M** and enter the values according to the table below.

SETTINGS	SUBDOMAIN 2
M	`M_pre*sin(k*x) 0 M_pre*cos(k*x)`

This kind of premagnetization will result in a magnetic flux that only emerges on the lower side of the magnet. That is, on the side where the μ-metal plate is. Click **OK**.

Boundary Conditions

Along the exterior boundaries, the magnetic field should be tangential to the boundary as the flow lines should form closed loops around the magnet. The natural boundary condition from the equation is

$$\mathbf{n} \cdot (\mu_0 \nabla V_m - \mu_0 \mathbf{M}) = \mathbf{n} \cdot \mathbf{B} = 0$$

Thus the magnetic field is made tangential to the boundary by a Neumann condition on the potential. On the interior boundary representing the μ-metal plate, you apply a special boundary condition for thin sheets of highly permeable material. Such plates are often used for the purpose of magnetic shielding.

Enter these boundary conditions according to the following table. To access interior boundaries, select the **Interior boundaries** check box.

SETTINGS	BOUNDARIES 1–4, 10, 11, 13, 14	BOUNDARY 5	BOUNDARIES 6–9, 12, 15
Type	Magnetic insulation	Magnetic shielding	Continuity
Relative permeability		`mu_r`	
Thickness		`th`	

Point Settings

So far the magnetic potential is not constrained anywhere and the problem has infinite number of solutions. Thus, you must supply a condition fixing it to a specific value somewhere. In the **Point Settings** dialog box, constrain the **Magnetic potential** to zero at Point 1.

MESH GENERATION

Click the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

There is no need to change the solver settings. The default solver settings are the conjugate gradients solver and the algebraic multigrid preconditioner. This is normally the most efficient solver setting for Poisson's equation, which is the type of equation that this model solves. This problem generates symmetric matrices so this option is selected as default. Click the **Solve** button on the Main toolbar. The computation time is about 6 seconds on a 3 GHz P4 machine.

POSTPROCESSING AND VISUALIZATION

The default plot shows a slice plot of the magnetic potential. To reproduce the plot in Figure 5-4 on page 231, follow these steps:

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

2 On the **General** page, select the **Slice** and **Arrow** check boxes in the **Plot type** area.

3 Click the **Slice** tab. In the **Predefined quantities** list, select **Magnetic flux density, norm**. Set the **Slice positioning x levels** to `0`, set **y levels** to `1`, and set **z levels** to `0`.

4 Click the **Arrow** tab. From the **Predefined quantities** list on the **Subdomain Data** page, select **Magnetic flux density**.

5 In the **Arrow positioning** area, set the numbers of **x points** to `50`, **y points** to `1`, and **z points** to `50`.

6 In the **Arrow parameters** area, select **Cone** from the **Arrow type** list.

7 Click **OK** to close the dialog box and generate the plot.

8 Use the **Zoom Window** tool on the Main toolbar to zoom in on the magnet and plate.

Magnetic Field in the Plate

The magnetic field in the plate is not available as a predefined postprocessing variable. To make this quantity more readily available, define extra expressions that you can use for postprocessing:

1 From the **Options** menu, select **Expressions>Boundary Expressions**.

2 Specify the variable names and expressions on Boundary 5 (the plate) according to the following table.

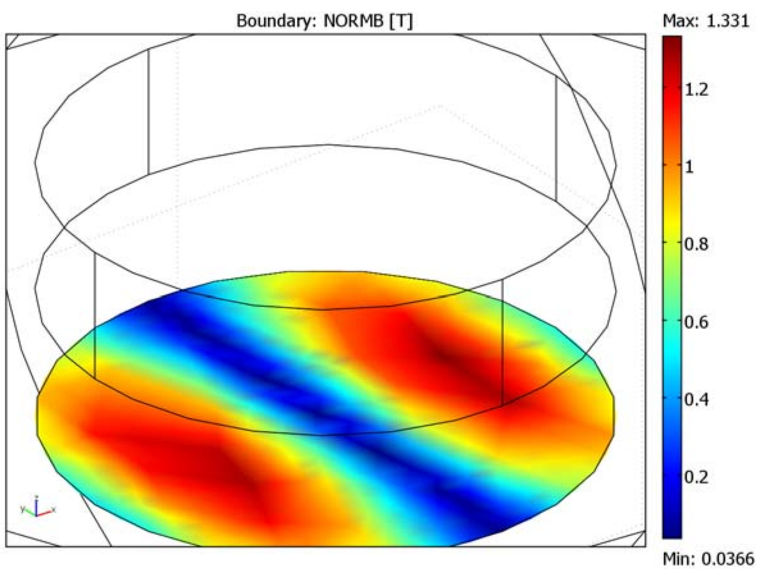
NAME	EXPRESSION
NORMH	sqrt(VmTx*VmTx+VmTy*VmTy+VmTz*VmTz)
NORMB	mu0_emnc*mu_r*NORMH

3 Click **OK** to close the dialog box.

4 From the **Solve** menu, select **Update Model** to compute values for the newly defined variables using the existing solution.

It is now possible to display the magnetic flux density in the plate using the available postprocessing functionality:

1 Select **Plot Parameters** from the **Postprocessing** menu or click the **Plot Parameters** button on the Main toolbar to open the **Plot Parameters** dialog box.
2 In the **Plot type** area on the **General** page, clear the **Slice** and **Arrow** check boxes and select the **Boundary** check box.
3 On the **Boundary** page, set the **Expression** to `NORMB`.
4 Click **OK** to generate the plot.



Force Calculation

To calculate the force on the plate, use the surface stress tensor

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

where $\mathbf{n}_1$ is the boundary normal pointing out from the plate and T_2 the stress tensor for air.

In this model the **H** and **B** fields are discontinuous across the plate, which makes it necessary to evaluate the fields on both sides of the plate.

All surfaces have an up and a down side. To find out which side is up and which is down, you can make an arrow plot of the components of the up normal (`unx`, `uny`, `unz`) on the boundaries. The up normal is the outward normal from the up side. In this case, you evaluate the fields on both sides and there is no need to know the direction of the up normal. The application mode defines variables for the surface stress tensor on the up and down side of the boundaries, for example, `unTz_emnc` and `dnTz_emnc` for the z component of the surface stress tensor.

Now, the x and y components of the force vanish for symmetry reasons. To find the force in the z direction, integrate the surface stress tensor over the plate. In the **Boundary Integration** dialog box, select Boundary 5 and enter the **Expression** `unTz_emnc+dnTz_emnc`. Click **OK** to evaluate the integral, which gives 1.2 N. This result is displayed in the message log at the very bottom of the user interface.

Force Calculation with Uniform Magnetization

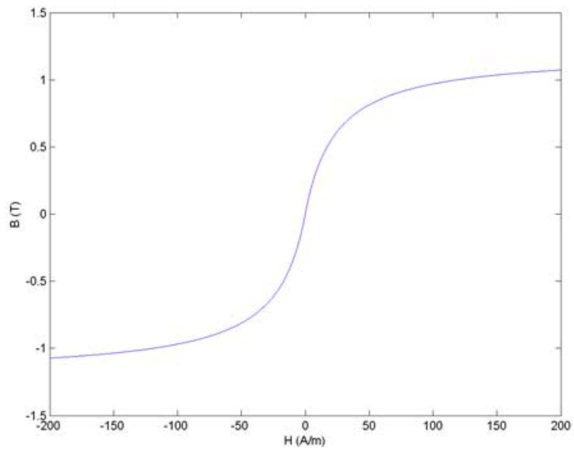
In the **Constants** dialog box change the constant `k` to `0` in order to obtain a uniform magnetization of $(0, 0, M_{\mathrm{pre}})$. Solve and calculate the force on the plate again. What is the conclusion?

Modeling Using a Nonlinear Magnetic Material in the Plate

Magnetic saturation effects are important in many applications. This step of the modeling process shows how to include a nonlinear magnetic material with saturation in the plate. The relative permeability μ_r is a function of the magnitude, $|\mathbf{H}|$ of the magnetic field as described by the following equation:

$$\mu_r = 1 + \left[(\mu_{\mathrm{rmax}} - 1)^{-1} + \left(\frac{\mu_0|\mathbf{H}|}{B_{\mathrm{sat}}}\right)^{\frac{1}{n}}\right]^{-1}$$

B_{sat} is the magnetic saturation flux, which for μ-metal is about 1.2 T.



Before modifying the model, we recommend that you save it under a new name. You do this from the **File** menu by selecting **Save As**.

OPTIONS AND SETTINGS

1 In the **Constants** dialog box enter the following names, expressions, and descriptions (the descriptions are optional).

NAME	EXPRESSION	DESCRIPTION
k	pi/10[mm]	Wave number in x direction
M_pre	5e5[A/m]	Magnetization amplitude in magnet
mu_rmax	4e4	Maximum relative permeability in plate
th	0.5[mm]	Plate thickness
B_sat	1.2[T]	Saturation flux density
N	1	Model order parameter

2 Click **OK** to close the dialog box.

Expression variables are used for the nonlinear permeability.

From the **Options** menu, select **Expressions>Boundary Expressions**.

1 Specify the variable names and expressions on Boundary 5 (plate) according to the following table.

NAME	EXPRESSION
`NORMH`	`sqrt(VmTx*VmTx+VmTy*VmTy+VmTz*VmTz)`
`NORMB`	`mu0_emnc*mu_r*NORMH`
`mu_r`	`1-1/(1/(1-mu_rmax)-(mu0_emnc*NORMH/B_sat)^(1/N))`

2 Click **OK** to close the dialog box.

PHYSICS SETTINGS

Initial Conditions

The nonlinear solver works by performing iterations, linearizing and solving the model for a sequence of operating points that converges to the final solution. For this to work properly in the first iteration, you need to start using an initial guess with a nonzero gradient:

1 From the **Physics** menu, open the **Subdomain Settings** dialog box.

2 Click the **Init** tab. Select all subdomains in the **Subdomain selection** list.

3 In the **Vm(t_0)** edit field, type `1[A/m]*sqrt(x^2+y^2+z^2)`.

4 Click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

The computation time is about 35 seconds on a 3 GHz P4 machine.

POSTPROCESSING AND VISUALIZATION

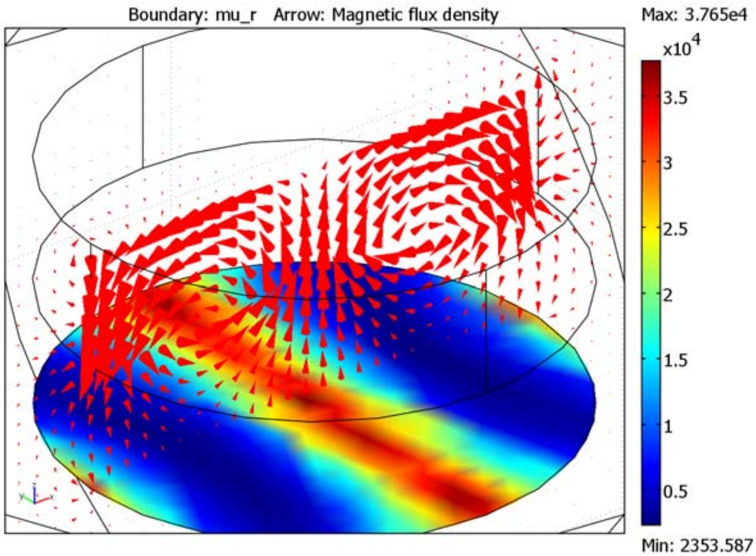
Plot the relative permeability in the plate and the magnetic flux density using the available postprocessing functionality:

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

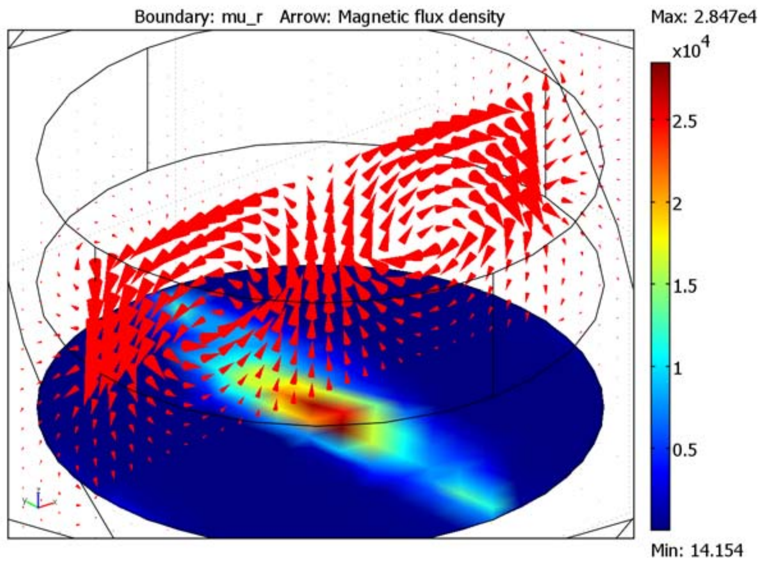
2 On the **General** page, select the **Boundary** and **Arrow** check boxes in the **Plot type** area.

3 On the **Boundary** page, set the **Expression** to `mu_r`.

4 For the **Arrow** plot, use the same settings as when generating Figure 5-4. Thus, click **OK** to close the dialog box and generate the plot.


Boundary: mu_r Arrow: Magnetic flux density
Max: 3.765e4
x10^4
3.5
3
2.5
2
1.5
1
0.5
Min: 2353.587

The plate has begun to saturate as the relative permeability is well below $4 \cdot 10^4$ in the blue areas. Try increasing the value of the premagnetization `M_pre` to `1e6` in the **Constants** dialog box and solve again to see a more pronounced saturation effect.



The computation time is about 100 seconds on a 3 GHz P4 machine.

Force Calculation

In the **Boundary Integration** dialog box, select Boundary 5 and enter the **Expression** `unTz_emnc+dnTz_emnc`. Click **OK** to evaluate the integral, which gives 3.3 N. This result appears in the message log at the bottom of the COMSOL Multiphysics user interface.

Magnetic Signature of a Submarine

Introduction

A vessel traveling on the surface or under water gives rise to detectable local disturbances in the Earth's magnetic field. These disturbances can be used to trigger weapon systems. The magnetic signature of a ship can be reduced by generating a counteracting magnetic field of suitable strength and direction based on prior knowledge of the magnetic properties of the vessel. An important step in the design of a naval ship is therefore to predict its magnetic signature. Another application where magnetic signatures are of great importance is in urban traffic control. Magnetic sensors, buried in our streets, are used to sense vehicles and control traffic lights. Ships and cars are both to a large extent made of sheet metal. This makes them hard to simulate using standard finite element analysis because volume meshes of thin extended structures are difficult to generate and tend to become very large. This model demonstrates a powerful technique that circumvents the problem by modeling the sheet metal as 2D faces embedded in a 3D geometry. Thus it is only necessary to create comparatively inexpensive 2D face meshes in addition to the 3D volume mesh used for the surrounding medium. A tangential projection of the 3D equation is then solved on the 2D face mesh.

Figure 5-5: Submarine HMAS Collins, image courtesy of Kockums AB.

Model Definition

In magnetostatic problems, where no currents are present, the problem can be solved using a scalar magnetic potential. A special technique to model thin sheets of high permeability materials is also demonstrated. The model geometry is shown in Figure 5-6. It consists of face objects representing the submarine enclosed in a 3D box representing the surrounding water.

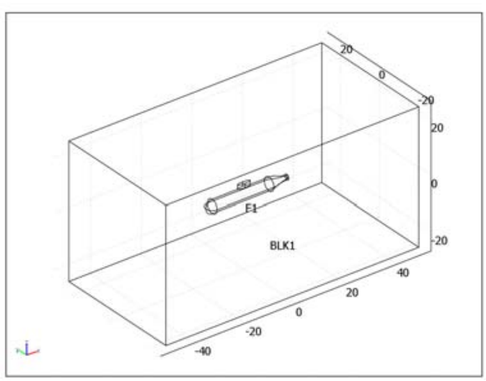


Figure 5-6: The model geometry.

DOMAIN EQUATIONS

In a current-free region, where

$$\nabla \times \mathbf{H} = \mathbf{0}$$

it is possible to define the scalar magnetic potential, V_m, from the relation

$$\mathbf{H} = -\nabla V_m$$

This is analogous to the definition of the electric potential for static electric fields. Using the constitutive relation between the magnetic flux density and magnetic field

$$\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$$

together with the equation

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

you can derive an equation for V_m,

$$-\nabla \cdot (\mu_0 \nabla V_m - \mu_0 \mathbf{M}) = 0$$

BOUNDARY CONDITIONS

Along the vertical exterior boundaries of the surrounding box, the magnetic field is assumed to be tangential to the boundaries. The natural boundary condition from the equation is

$$\mathbf{n} \cdot (\mu_0 \nabla V_m - \mu_0 \mathbf{M}) = \mathbf{n} \cdot \mathbf{B} = 0$$

Thus the magnetic field is made tangential to the boundary by a Neumann condition on the potential. In order to emulate the Earth's magnetic field a potential condition is applied on the top boundary

$$V_m = 1989(A)$$

whereas the bottom boundary is set to a magnetic potential of zero.

$$V_m = 0$$

This corresponds to a vertical magnetic flux density of about 0.5 G—a typical value for the Earth's magnetic field. On the face objects representing the hull of the submarine, you apply a 2D tangential projection of the 3D domain equation where the thickness and permeability of the hull are introduced as parameters. This is readily available in the used formulation as a shielding boundary condition, which is useful for modeling of highly permeable thin sheets. Corresponding boundary conditions are available in the Conductive Media DC and Electrostatics application modes for modeling of thin sheets with high conductance and high permittivity respectively.

Results and Discussion

Figure 5-7 shows the magnetic flux density in a horizontal slice plot 7.5 m below the keel of the submarine. A distinct field perturbation due to the presence of the vessel can be seen. The magnitude and direction of the tangential magnetic field in the hull of the vessel is shown using color and arrows.

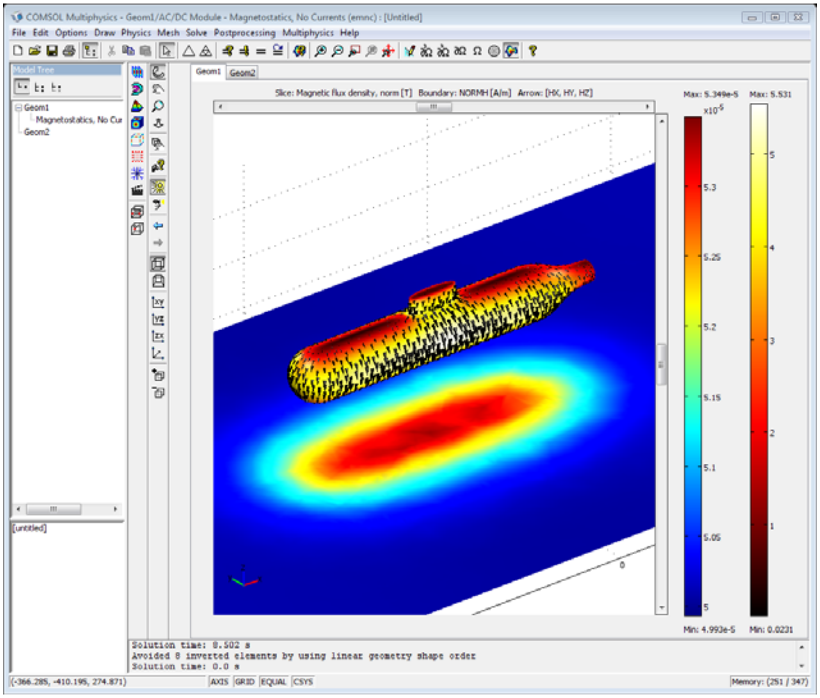


Figure 5-7: The slice color plot shows the magnetic flux density. The arrows and boundary color plot show the direction and strength of the tangential magnetic field in the hull.

Model Library path: `ACDC_Module/General_Industrial_Applications/submarine`

Modeling Using the Graphical User Interface

Select the **3D>AC/DC Module>Statics>Magnetostatics, No Currents** application mode in the **Model Navigator**. Click **OK**.

Open the **Work-Plane Settings** dialog box from the **Draw** menu, use the default *xy* work plane and click **OK**.

OPTIONS AND SETTINGS

1 On the **Grid** tab in the **Axes/Grid Settings** dialog box, clear the **Auto** check box and enter the grid settings according to the following table.

GRID	
x spacing	`0.5`
y spacing	`0.5`

GEOMETRY MODELING

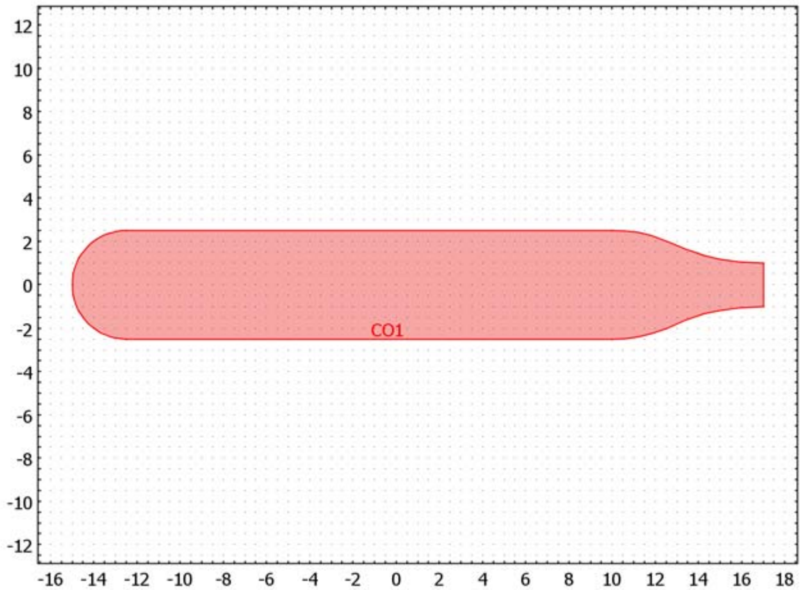
1 Click the **Rectangle** button on the Draw toolbar to create a rectangle with its lower left corner at (–12.5, –2.5) with a width of 22.5 and a height of 5.

2 Click the **Ellipse/Circle (Centered)** button on the Draw toolbar to create a circle with the radius 2.5 and its center at (–12.5, 0).

3 Click the **3rd Degree Bézier Curve** button on the Draw toolbar and then click in sequence at the following points (10.0, 2.5), (13.0, 2.5), (13.0, 1.0) and (17.0, 1.0). Then, select the **Line** tool from the toolbar and click at (17.0, -1.0). Finally, go back to the **3rd Degree Bézier Curve** tool and click at (13.0, -1.0), (13.0, -2.5) and (10.0, -2.5). Close the curve by clicking the right mouse button.

4 Press Ctrl+A to select all objects and then click the **Union** button on the toolbar.

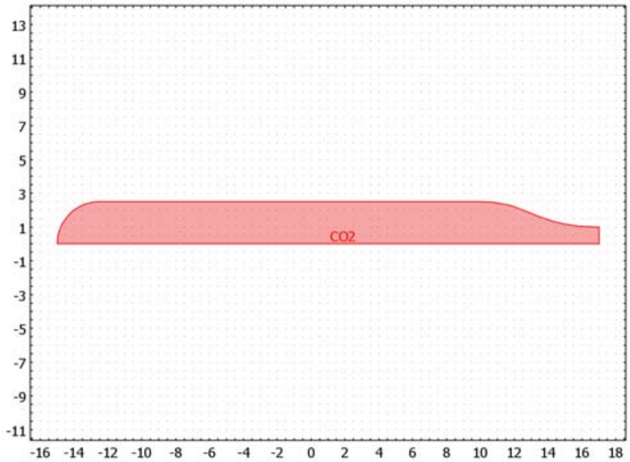
5 Click the **Delete Interior Boundaries** button on the Draw toolbar. Then click the **Zoom Extents** button.



6 Click the **Rectangle** button on the Draw toolbar to create a rectangle with its lower left corner at (–15, –2.5), with a width of 32 and a height of 2.5.

7 Press Ctrl+A to select all objects and then click the **Difference** button on the Draw toolbar.

8 Click the **Zoom Extents** button.



9 Select the composite object CO2.

10 Open the **Revolve** dialog box from the **Draw** menu. Set the coordinates for the **Second point** to (1, 0) and click **OK** to revolve into the 3D geometry.

11 Go back to the work plane **Geom2**.

12 Select **Ellipse/Circle (Centered)** from the **Draw** menu and create an ellipse with the **A-semiaxis** 2.5 and the **B-semiaxis** 0.625 and its center at (0, 0).

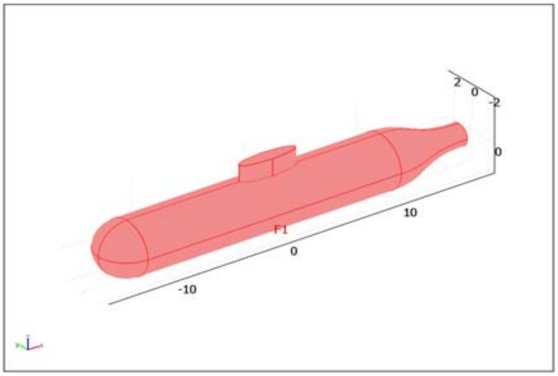
13 With the ellipse selected, open the **Extrude** dialog box from the **Draw** menu and extrude it a **Distance** of 2 m.

14 In the 3D view, click the **Move** button in the Draw toolbar to move the extruded object a distance of 2 m in the z direction.

15 Press Ctrl+A to select all objects and then click the **Union** button on the toolbar.

16 Click the **Delete Interior Boundaries** button on the Draw toolbar.

17 Click the **Coerce to Face** button on the toolbar. Then click the **Zoom Extents** button



18 Click the **Block** button on the Draw toolbar, set the **Base** to **Center**, the **Axis base point** to (0, 0, 0), and the **Length** to (100, 50, 50); when done, click **OK**.

PHYSICS SETTINGS

Subdomain Settings

Keep the default subdomain settings.

Boundary Conditions

Along the vertical boundaries of the surrounding box, the magnetic field should be tangential to the boundaries. In order to emulate the Earth's magnetic field, apply a potential condition of 1989 A on the top boundary. At the bottom boundary, set a magnetic potential of zero. On the interior boundaries representing the hull of the submarine, apply a special boundary condition denoted shielding. This makes use of a 2D tangential projection of the 3D domain equation where the thickness and permeability of the hull are introduced as parameters.

Enter the boundary conditions according to the following table. In order to access the interior boundaries, select the **Interior boundaries** check box.

SETTINGS	BOUNDARIES 1, 2, 5, 27	BOUNDARIES 6–26	BOUNDARY 4	BOUNDARY 3
Boundary condition	Magnetic insulation	Magnetic shielding	Magnetic potential	Zero potential
μ_r		700		
d		0.05		
V_{m0}			1989	

MESH GENERATION

1 Open the **Free Mesh Parameters** dialog box.
2 Click the **Custom mesh size** button. Type `0.5` in the field **Maximum element size scaling factor**, and type `0.4` in the **Mesh curvature factor** edit field. This limits the maximum mesh size to one half of the default value and provides a higher resolution of the curvature than the default settings for a normal 3D mesh.
3 Click the **Initialize Mesh** button; then click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar. This starts the analysis using the default solver, the Conjugate gradients solver, and the default preconditioner, Algebraic multigrid.

The computation time is about 1 minute on a 3 GHz P4 machine.

POSTPROCESSING AND VISUALIZATION

The default plot shows a slice plot of the magnetic potential with 5 slices perpendicular to the x axis. A more interesting plot can easily be obtained.

1 Select **Plot Parameters** from the **Postprocessing** menu to open the **Plot Parameters** dialog box and click the **Slice** tab.
2 Select **Magnetic flux density, norm** as **Expression**.
3 Set the numbers of **x-levels** and **y-levels** to `0`.
4 For z, select **Vector with coordinates** and set the value to `-10`. Click **OK** to close the dialog and make the plot.

Tangential Magnetic Field in the Hull

The magnetic field in is available as predefined postprocessing variables available everywhere. Do display variables only on some boundaries, it is possible to use the **Suppress>Boundaries** option from the **Options** menu. Another convenient way to select boundaries used in a plot, is to define a new boundary variable that only has valid expressions on the boundaries used in the plot.

From the **Options** menu, select **Expressions>Boundary Expressions**.

1 Specify the variable names and expressions on the hull Boundaries 6–26, according to the following table.

NAME	EXPRESSION
NORMH	normtH_emnc
HX	tHx_emnc
HY	tHy_emnc
HZ	tHz_emnc

2 Click **OK** to close the dialog box.

3 From the **Solve** menu, select **Update Model** in order to compute values for the newly defined variables using the existing solution.

The magnetic field in the hull can now be displayed using the available postprocessing functionality in COMSOL Multiphysics.

1 Select **Plot Parameters** from the **Postprocessing** menu to open the **Plot Parameters** dialog box.

2 On the **General** page, select the **Slice**, **Boundary**, and **Arrow** check boxes in the **Plot type** area.

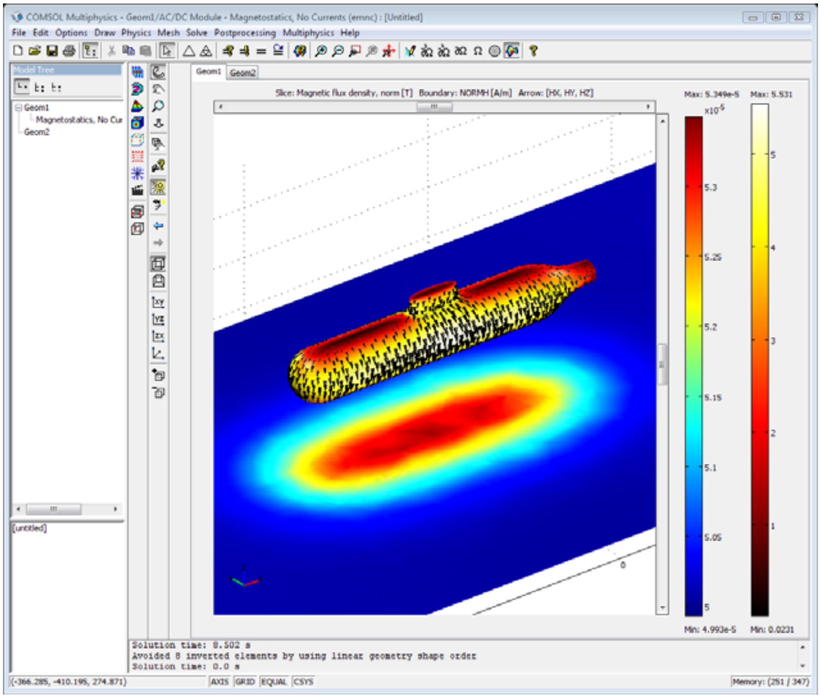
3 On the **Slice** page, keep the previous settings.

4 On the **Boundary** page, type NORMH in the **Expression** edit field.

5 On the **Arrow** page, select **Boundaries** from the **Plot arrows on** list. On the **Boundary Data** page, enter HX as the **x component**, HY as the **y component**, and HZ as the **z component**.

6 In the **Arrow parameters** area, clear the **Auto** check box, then specify a **Scale factor** of 0.2.

7 Click **OK** to generate the plot.



3D Quadrupole Lens

Introduction

This is a full 3D version of the Quadrupole Lens model in the COMSOL Multiphysics Model Library. For a general overview of the setup and the physics, see "Quadrupole Lens" on page 83 in the *COMSOL Multiphysics Model Library.* This version of the model also takes fringing fields into account, and the calculation of the forces on the ions uses all components of their velocities.

Model Definition

The geometry of the quadrupole lens is the same as outlined in the documentation to the 2D model: three quadrupoles in a row, followed by 1 m of empty space, where the ions are left to drift. The AC/DC Module features an application mode for magnetostatics without currents. This reduces the memory usage considerably compared to the formulation including currents.

DOMAIN EQUATIONS

The magnetic field is described using the Magnetostatics equation, solving for the magnetic scalar potential V_m (Wb/m):

$$-\nabla\cdot(\mu_0\nabla V_m-\mu_0\mathbf{M}) = 0$$

where $\mu_0 = 4\pi\cdot10^{-7}$ H/m denotes the permeability of vacuum and $\mathbf{M}$ is the magnetization (A/m). In the iron subdomain

$$-\nabla\cdot(\mu_0\mu_r\nabla V_m) = 0$$

where $\mu_r = 4000$ is the relative permeability. The magnetic scalar potential is everywhere defined so that $\mathbf{H} = -\nabla V_m$.

BOUNDARY CONDITIONS

The *magnetic insulation* boundary condition, reading $\mathbf{n}\cdot\mathbf{B} = 0$, is used all around the iron cylinder, and at the lateral surfaces of the air domain that encloses the drift length. At the base surface of the same air domain, the magnetic potential is zero.

Results

The magnetic field density and arrows showing its direction are depicted in the figure below.

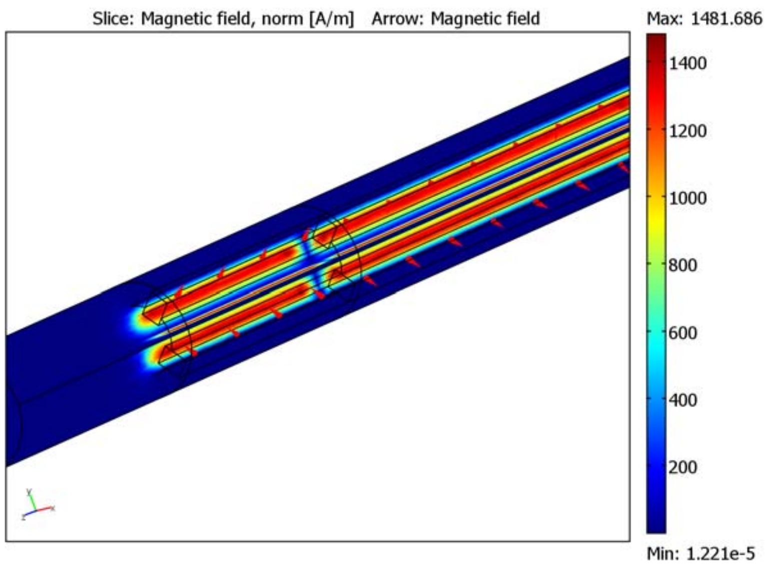


Figure 5-8: Slice plot and arrows of the magnetic field density in the quadrupole lens.

Each ion passing through the assembly experiences Maxwell forces equal to $\mathbf{F} = q\mathbf{v}\times\mathbf{B}$, where $\mathbf{v}$ (m/s) is the velocity of the ion. To find the transverse position as a function of time, solve Newton's second law for each ion: $q\mathbf{v}\times\mathbf{B} = m\mathbf{a}$. Figure 5-9 shows the traces of the ions as they fly through the quadrupole lens.

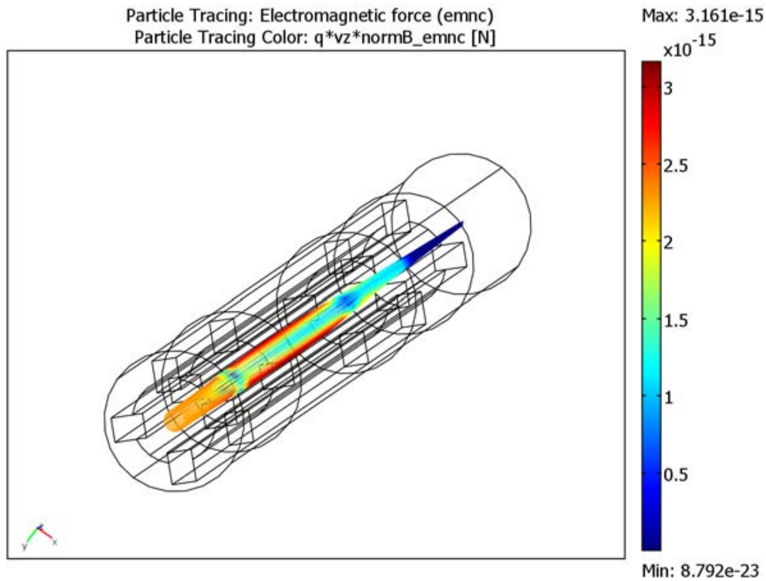


Figure 5-9: Particle tracing plot of the ions. The line colors show the local force acting on each ion. The force grows larger (red) far away from the center of the beam line and smaller (blue) where two oppositely polarized quadrupoles join.

Model Library path: `ACDC_Module/General_Industrial_Applications/quadrupole_3d`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator**, select **3D** in the **Space dimension** list.

2 Select **AC/DC Module>Statics>Magnetostatics, No Currents** from the list of application modes.

3 Click **OK** to close the dialog box.

OPTIONS AND SETTINGS

1 In the **Constants** dialog box, enter the following constant names, expressions, and (optionally) descriptions; when done, click **OK** to close the dialog box.

NAME	EXPRESSION	DESCRIPTION
`M`	`11`	`Ion mass number`
`Z`	`5`	`Ion charge number`
`vz`	`0.01*3e8[m/s]`	`Ion velocity`
`mp`	`1.672e-27[kg]`	`Proton mass`
`Ze`	`1.602e-19[C]`	`Proton charge`
`m`	`M*mp`	`Ion mass`
`q`	`Z*Ze`	`Ion charge`
`MQ`	`5.8e3[A/m]`	`Quadrupole magnetization`

GEOMETRY MODELING

1 Choose **Draw>Work-Plane Settings**. Click **OK** to get the default *xy*-plane.

2 Choose **Draw>Specify Objects>Rectangle**. Specify the following properties; when done, click **OK**.

PROPERTY	EXPRESSION
Width	`0.177`
Height	`0.07`
Position: Base	Corner
Position: x	`0`
Position: y	`-0.035`

3 Choose **Draw>Modify>Rotate**. Rotate the rectangle by 45 degrees around the origin. When done, click **OK**.

4 Choose **Draw>Specify Objects>Circle**. Specify the following properties, then click **OK**.

PROPERTY	EXPRESSION
Radius	`0.2`
Position: Base	Center
Position: x	`0.2`
Position: y	`0.2`

5 Press Ctrl+A to select both objects. Click the **Intersection** button on the Draw toolbar.

6 Press Ctrl+C to make a copy of the composed object (CO1), then press Ctrl+V to paste it at the same location, that is with zero displacements.

7 Rotate the copied object (CO2) by 90 degrees around the origin.

8 Paste two more copies of CO1 at the same location, and rotate them by 180 and 270 degrees, respectively.

9 Create a circle centered at the origin with a radius of 0.2 m.

10 Create another circle centered at the origin, with a radius of 0.12 m.

11 Click the **Create Composite Object** button. Create an object using the formula `C1+C2-(CO1+CO2+CO3+CO4)`. Click **OK**.

12 Once more, draw a circle centered at the origin with a radius of 0.2 m.

The geometry should now look like that in the figure below.

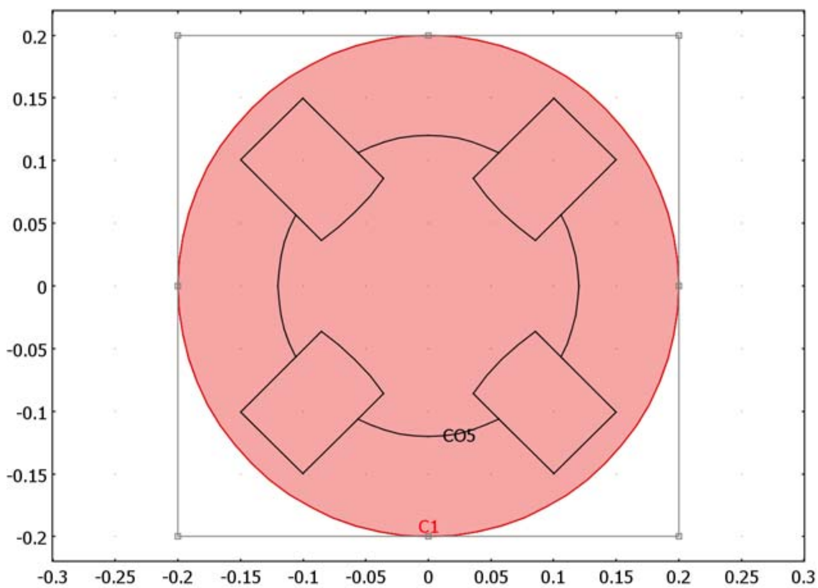


Figure 5-10: Work-plane view of the geometry.

13 Select all objects, then choose **Draw>Extrude**. Click **OK** to extrude the geometry to a height of 1 m.

14 In the 3D view, copy the object you just extruded, and paste it to the same location. You find the copy and paste commands in the **Edit** menu.

15 Choose **Draw>Modify>Scale**. Scale the pasted object by a factor of 2 in the z direction. When done, click **OK**.

16 Choose **Draw>Modify>Move**. Move the scaled object 1 m in the z direction. When done, click **OK**.

17 Paste another copy of the first object into the geometry, but now with a displacement of 3 m in the z direction.

18 Choose **Draw>Cylinder**. Create a cylinder with the following specifications; when done, click **OK**.

PROPERTY	EXPRESSION
Radius	`0.2`
Height	1
Axis base point z	4

PHYSICS SETTINGS

Subdomain Settings

1 In the **Subdomain Settings** dialog box, select Subdomains 1–3 and load **Iron** from the list of materials.

2 Leave the default settings in Subdomains 4 and 11–13.

3 In all the other subdomains, set the constitutive relation to $\mathbf{B} = \mu_0\mathbf{H} + \mu_0\mathbf{M}$ and set the magnetization according to the table below.

SETTINGS	SUBDOMAINS, 5, 7, 18	SUBDOMAINS 8, 10, 15	SUBDOMAINS, 9, 14, 16	SUBDOMAINS 6, 17, 19
M (x component)	`MQ/sqrt(2)`	`-MQ/sqrt(2)`	`MQ/sqrt(2)`	`-MQ/sqrt(2)`
M (y component)	`MQ/sqrt(2)`	`MQ/sqrt(2)`	`-MQ/sqrt(2)`	`-MQ/sqrt(2)`

Boundary Conditions

1 In the **Boundary Settings** dialog box, set the boundary conditions according to the table below. The easiest way to do this is to first select all the exterior boundaries, by selecting the **Select by group** check box and select one of them. Apply magnetic insulation to all the boundaries, and change the condition on Boundary 13 back to zero potential.

SETTING	BOUNDARY 13	ALL OTHERS
Boundary condition	Zero potential	Magnetic insulation

MESH GENERATION

1 In the **Free Mesh Parameters** dialog box, select **Finer** among the **Predefined mesh sizes**.
2 On the **Subdomain** tab, select Subdomains 4 and 11–13, and enter `0.03` for the **Maximum element size**.
3 On the **Advanced** page, set the **z-direction scale factor** to `0.2`.
4 Click the **Remesh** button. When the mesher has finished, click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows the magnetic potential. Follow the instructions to get an overview of the magnetic field.

1 In the **Plot Parameters** dialog box, go to the **Slice** page and select **Magnetic field, norm** from the **Predefined quantities** list. Set the **Number of levels** to (`1`, `1`, `0`).
2 On the **Arrow** page, select the **Arrow plot** check box. Set the **Number of points** to (`2`, `2`, `20`) and the **Scale factor** to `0.2`.
3 Click **Apply** to see the magnetic field.

To see how the ions travel through the system of quadrupoles, do the following:

1 On the **General** page, clear the **Slice** and **Arrow** check boxes, and select the **Particle tracing** check box.
2 On the **Particle tracing** page, select **Electromagnetic force** from the **Predefined forces** list. In the **Mass** edit field, type `m`.
3 On the **Start Points** page, type `0.03*cos(linspace(0,2*pi,41))` in the **x** edit field and `0.03*sin(linspace(0,2*pi,41))` in the **y** edit field.
4 On the **Initial Values** page, type `vz` in the rightmost **Initial velocity** edit field to set the initial z-velocity.
5 On the **Line Color** page, click the **Use expression** option button.
6 Click the **Color Expression** button to open the **Pathline Color Expression** dialog box. In the **Expression** edit field, type `q*vz*normB_emnc` (this is an approximate value of the local force acting on each ion). Click **OK**.
7 Click the **Advanced** button and set the **Relative tolerance** to `1e-6`.
8 Click **OK** to close the **Plot Parameters** dialog box and generate the particle tracing plot.

Superconducting Wire

Current can flow in a superconducting wire with practically zero resistance, although factors including temperature, current density, and magnetic field can limit this phenomenon. This model solves a time-dependent problem of a current building up in a superconducting wire close to the critical current density. This model is based on a suggestion by Dr. Roberto Brambilla, CESI - Superconductivity Dept., Milano, Italy.

Introduction

The Dutch physicist Heike Kamerlingh Onnes discovered superconductivity in 1911. He cooled mercury to the temperature of liquid helium (4 K) and observed that its resistivity suddenly disappeared. Research in superconductivity reached a peak during the 1980s in terms of activity and discoveries, especially when scientists uncovered the superconductivity of ceramics. In particular, it was during this decade that researchers discovered YBCO—a ceramic superconductor composed of yttrium, barium, copper, and oxygen with a critical temperature above the temperature of liquid nitrogen. However, researchers have not yet created a room-temperature superconductor, so much work remains for the broad commercialization of this area.

This model illustrates how current builds up in a cross section of a superconducting wire; it also shows where critical currents produce a swelling in the non-superconducting region.

Model Definition

The dependence of resistivity on the amount of current makes it difficult to solve the problem using the standard application modes in the AC/DC Module. The reason is this: A circular dependency arises because the current-density calculation contains the resistivity, leading to a resistivity that is dependent on itself.

An alternate approach uses the magnetic field as the dependent variable, and you can then calculate the current as

$$\mathbf{J} = \nabla \times \mathbf{H} .$$

The electric field is a function of the current, and Faraday's law determines the complete system as in

$$\nabla \times \mathbf{E}(\mathbf{J}) = -\mu \frac{\partial \mathbf{H}}{\partial t}$$

where $\mathbf{E}(\mathbf{J})$ is the current-dependent electric field. The model calculates this field with the empirical formula

$$\mathbf{E}(\mathbf{J}) = \begin{cases} \mathbf{0} & |\mathbf{J}| < J_C \\ E_0\left(\frac{|\mathbf{J}| - J_C}{J_C}\right)^{\alpha} \frac{\mathbf{J}}{|\mathbf{J}|} & |\mathbf{J}| \geq J_C \end{cases}$$

where E_0 and α are constants determining the nonlinear behavior of the transition to zero resistivity, and J_C is the critical current density, which decreases as temperature increases.

For the superconductor YBCO, this model uses the following parameter values (Ref. 1):

PARAMETER	VALUE
E_0	0.0836168 V/m
α	1.449621256
J_C	17 MA
T_C	92 K

Systems with two curl operators are best dealt with using vector elements (edge elements). This is the default element for the application modes in the AC/DC Module that solve similar equations. This particular formulation for the superconducting system is not available in the AC/DC Module, so you must define it using the PDE, General Form application mode. In addition, the model uses higher-order vector elements, called *curl elements* in COMSOL Multiphysics. The resulting system becomes

$$D_a \frac{d\mathbf{H}}{dt} + \nabla \cdot \Gamma = \mathbf{F} \Rightarrow \begin{bmatrix} \mu_0 & 0 \\ 0 & \mu_0 \end{bmatrix} \cdot \begin{bmatrix} \frac{dH_x}{dt} \\ \frac{dH_y}{dt} \end{bmatrix} + \nabla \cdot \begin{bmatrix} 0 & E_z(J_z) \\ -E_z(J_z) & 0 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}.$$

For symmetry reasons, the current density has only a z-component.

The model controls current through the wire with its outer boundary condition. Because Ampere's law must hold around the wire, a line integral around it must add up to the current through the wire. Cylindrical symmetry results in a known magnetic field at the outer boundary

$$\oint \mathbf{H} \cdot \mathbf{dl} = 2\pi r H_\phi = I_{\mathrm{wire}} \Rightarrow H_\phi = \frac{I_{\mathrm{wire}}}{2\pi r}.$$

For vector elements, the Dirichlet boundary conditions add a constraint on the tangential component of the vector field. If the field components are called H_x and H_y, the tangential counterparts are $\mathrm{t}H_x$ and $\mathrm{t}H_y$.

Results and Discussion

The model applies a simple transient exponential function as the current through the wire, reaching a final value of 1 MA. This extremely large current is necessary if the superconducting wire is to reach its critical current density. Figure 5-11 plots the current density at two times after the current starts.

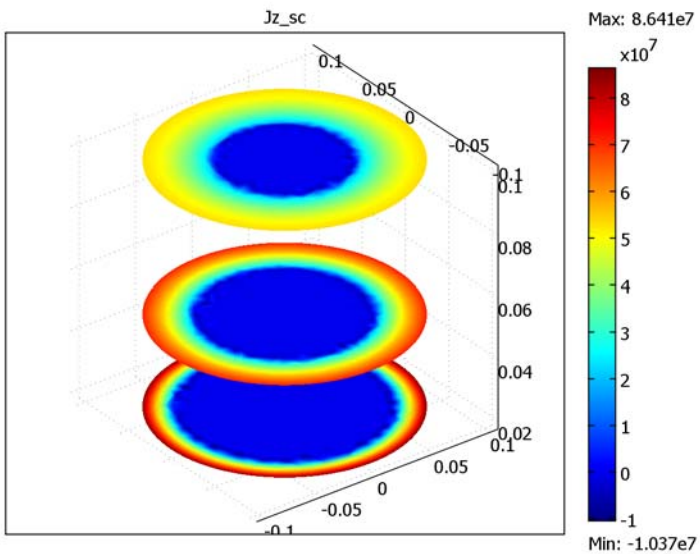


Figure 5-11: The current density at t = 0.02, 0.05, and 0.1. The swelling of the ring comes from the transition out of the superconducting state at current densities exceeding J_C.

Reference

1. R. Pecher, M.D. McCulloch, S.J. Chapman, L. Prigozhin, and C.M. Elliotth, "3D-modelling of bulk type-II superconductors using unconstrained H-formulation," *6th European Conf. Applied Superconductivity* (EUCAS 2003).

Model Library path: `ACDC_Module/General_Industrial_Applications/superconducting_wire`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. In the **Model Navigator**, select **2D** from the **Space dimension** list.
2. Select the **COMSOL Multiphysics>PDE Modes>PDE, General Form>Time-dependent analysis** application mode.
3. In the **Dependent variables** edit field, type `Hx Hy`. Click **OK**.

OPTIONS AND SETTINGS

Because the PDE modes do not support units, disable COMSOL Multiphysics' unit handling:

1. From the **Physics** menu, choose **Model Settings**.
2. From the **Base unit system** list, select **None**, then click **OK**.

Next, proceed with defining the model.

1. From the **Options** menu, choose **Constants**.
2. In the **Constants** dialog box, define the following constants with names, expressions, and descriptions (optional):

NAME	EXPRESSION	DESCRIPTION
`mu0`	`4*pi*1e-7`	`Permeability of vacuum (H/m)`
`alpha`	`1.449621256`	`Parameter for resistivity model`
`Jc`	`1.7e7`	`Critical current density (A/m^2)`
`I0`	`1e6`	`Applied current (A)`
`rho_air`	`1e6`	`Resistivity of air (ohm*m)`
`tau`	`0.02`	`Time constant for applied current (s)`

NAME	EXPRESSION	DESCRIPTION
Tc	92	Critical temperature (K)
dT	4	Parameter for resistivity model (K)
dJ	Jc/1e4	Parameter for resistivity model (A/m^2)
E0	0.0836168	Parameter for resistivity model (V/m)

3 Click **OK**.

GEOMETRY MODELING

You can access all the dialog boxes for specifying primitive objects from the **Draw** menu under **Specify Objects**. The first column in the table below contains the labels of the geometric objects. These are automatically generated by COMSOL Multiphysics, and you do not have to enter them. Just check that you get the correct label for the objects that you create.

1 Draw two circles with properties according to the table below.

LABEL	RADIUS	BASE	BASE, X	BASE, Y
C1	0.1	Center	0	0
C2	1	Center	0	0

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

PHYSICS SETTINGS

1 From the **Options** menu, choose **Expressions>Scalar Expressions**.

2 In the **Scalar Expressions** dialog box, define the following variables with names, expressions, and (optionally) descriptions:

NAME	VALUE	DESCRIPTION
Jz_sc	HyxHxy	Current density (A/m^2)
normH_sc	sqrt(Hx^2+Hy^2)	Norm of the H-field (H/m)
I1	I0*(1-exp(-t/tau))	Applied current (A)
Q_sc	Ez_sc*Jz_sc	Generated heat in superconductor (W/m^3)

The expression HyxHxy is a predefined short form for Hyx-Hxy, the z-component of the curl of the **H**-field:

$$\frac{\partial H_y}{\partial x} - \frac{\partial H_x}{\partial y}$$

3 Click **OK**.

4 From the **Options** menu, choose **Expressions>Boundary Expressions**.

5 In the **Boundary Expressions** dialog box, define the following variable:

NAME	EXPRESSION ON BOUNDARIES 1, 2, 5, 8	ALL OTHERS
H0phi	I1/(2*pi*sqrt(x^2+y^2))	

6 Click **OK**.

7 From the **Options** menu, choose **Expressions>Subdomain Expressions**.

8 In the **Subdomain Expressions** dialog box, define the following variable:

NAME	EXPRESSION IN SUBDOMAIN 1	EXPRESSION IN SUBDOMAIN 2
Ez_sc	rho_air*Jz_sc	E0*((Jz_sc-Jc)* flc2hs(Jz_sc-Jc-dJ,dJ)/Jc)^alpha

Boundary Conditions

1 From the **Physics** menu, open the **Boundary Settings** dialog box.

2 Enter boundary conditions according to the following table:

SETTINGS	BOUNDARIES 2, 8	BOUNDARIES 1, 5
Boundary condition	Dirichlet	Dirichlet
R_1	H0phi*t1x+tHx	H0phi*t1x-tHx
R_2	H0phi*t1y+tHy	H0phi*t1y-tHy

3 Click **OK**.

Subdomain Settings

1 From the **Physics** menu, open the **Subdomain Settings** dialog box.

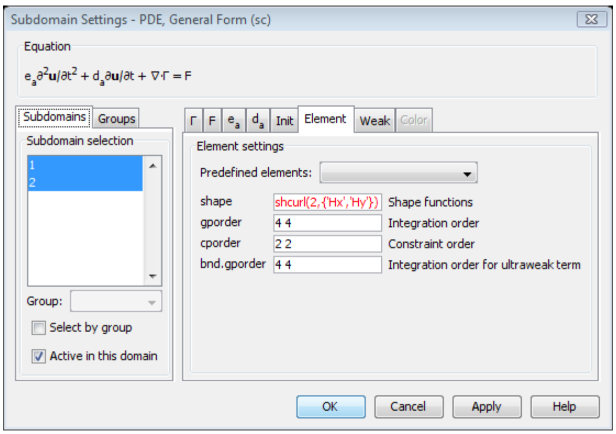
2 Specify subdomain settings according to the following table:

SETTINGS	SUBDOMAINS 1, 2
Γ_{11}	0
Γ_{12}	Ez_sc
Γ_{21}	-Ez_sc
Γ_{22}	0
F_1	0
F_2	0
$D_{a,11}$	mu0

SETTINGS	SUBDOMAINS 1, 2
$D_{a,12}$	0
$D_{a,21}$	0
$D_{a,22}$	`mu0`

3 Click the **Element** tab.

4 Make sure that both subdomains are selected, then type `shcurl(2,{'Hx','Hy'})` in the **shape** edit field to use second-order vector elements when solving the model.



The shape function entry turns red when you enter a shape function using the syntax with curly braces, but you can disregard this indication.

5 Click **OK**.

MESH GENERATION

1 From the **Mesh** menu, choose **Free Mesh Parameters**.

2 From the **Predefined mesh sizes** list, select **Coarse**.

3 Click the **Subdomain** tab. Select Subdomain 2, then type `2e-2` in the **Maximum element size** edit field.

4 Click the **Remesh** button, then click **OK**.

COMPUTING THE SOLUTION

1 From the **Solve** menu, choose **Solver Parameters**.

2 In the **Times** edit field, type `linspace(0,0.1,21)`.

3 On the **Time Stepping** page, select the **Manual tuning of step size** check box. In the **Initial time step** edit field, type `1e-9`. In the **Maximum time step** edit field, type `1e-3`.

4 Click **OK**.

5 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

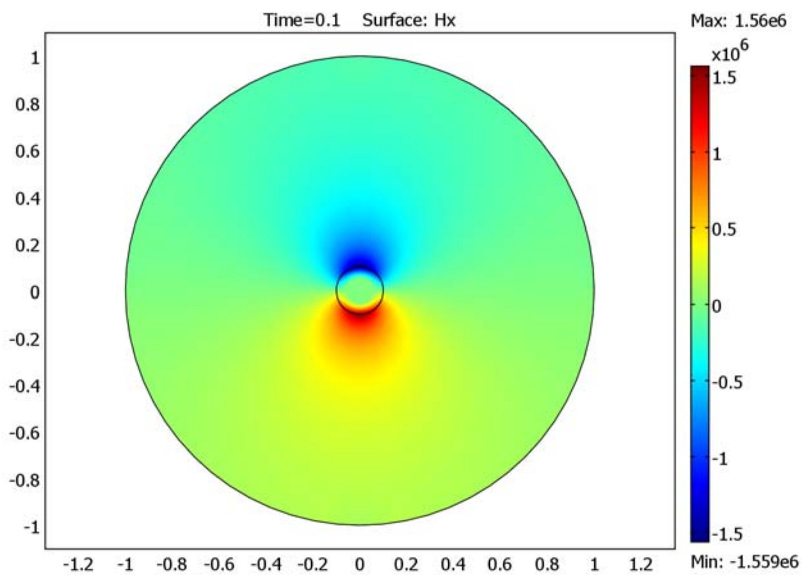
1 From the **Postprocessing** menu, select **Plot Parameters**.

2 On the **General** page, make sure that the **Surface** and **Geometry edges** check boxes are selected in the **Plot type** area.

3 On the **Surface** page, verify that `Hx` is in the **Expression** edit field.

4 Click **OK**.

You should now see the following plot in the drawing area on your screen.



5 From the **Postprocessing** menu, select **Domain Plot Parameters**.

6 On the **General** page, select the solutions at times **0.02**, **0.05**, and **0.1**.

7 Click the **Surface** tab. From the **Subdomain selection** list, select Subdomain 2. In the **Expression** edit field, type `Jz_sc`.

8 Click **OK**.

You should now see the plot shown in Figure 5-11 on page 261.

RF-Heated Hot Wall Furnace for Semiconductor Processing

Introduction

Furnace reactors are used in semiconductor fabrication to grow layers of semiconductor materials on wafers. An important aspect is to grow the layers with a crystal structure that matches that of the wafer substrate. This is called epitaxial growth and is a key technology for the fabrication of electronic devices. For silicon carbide (a wide-bandgap semiconductor), epitaxial growth is one of the great challenges towards manufacturing reliable devices.

The layer growth takes place with the wafers sitting in graphite susceptors (wafer holders) at very high temperatures in the range of 2000 °C. The susceptors are heated with radio-frequency (RF) coils at power levels in the 10 kW range. The design of the reactor chamber is crucial for maintaining a uniform temperature, efficient heating, and control of high-temperature regions. It is especially important that the quartz tube surrounding the susceptor remains at moderate temperatures.

This COMSOL Multiphysics model examines a simple furnace design that heats a graphite susceptor using an 8-kW RF signal at 20 kHz. It determines the temperature distribution over the wafer along with the temperature on the outer quartz tube. At these high temperatures the heat flux is dominated by radiation. The model requires the AC/DC Module and the Heat Transfer Module.

Model Definition

Figure 5-12 shows the furnace geometry. The quartz tube is surrounded by six coils that carry the RF power. The alternating field induces currents in the conductive center of the graphite susceptor, and these currents heat that wafer holder. The wafer is placed at the bottom of a chamber located inside the susceptor, where almost the entire bottom is made of silicon carbide to avoid contamination. The outer part of the susceptor is made of graphite foam, which serves as thermal insulation.

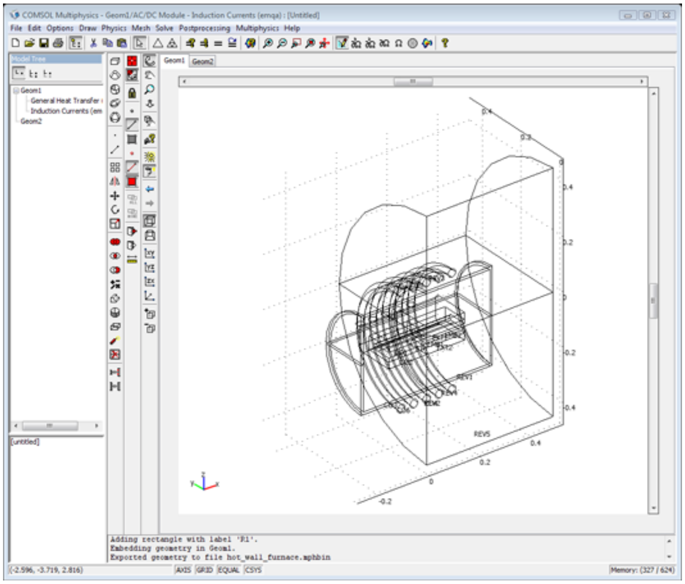


Figure 5-12: The furnace geometry cut in half. Note that you need model only one quarter of the full geometry due to symmetry.

Thanks to symmetry, you can twice divide the furnace in half and thus need include only a quarter of its geometry in the simulation.

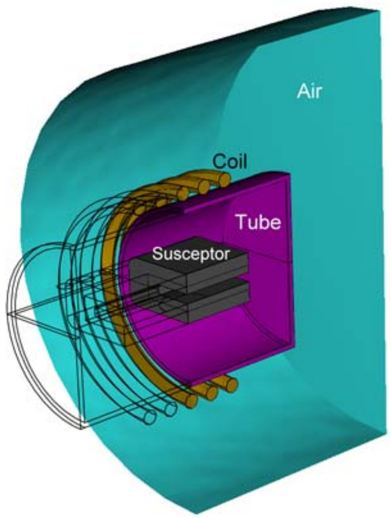


Figure 5-13: The part of the geometry included in the simulation, also with the surrounding air domain.

The coils are made of copper, and at the frequency of interest the skin depth is in the order of 0.5 mm, so it is possible to model the current as a surface current. In fact, because the center core of the coils does not conduct current, they are actually constructed as tubes. These surface currents make it possible to use one of the AC/DC Module's induction-current application modes, which solves only for the **A**-field. The total current used in this case is 21.5 kA, which corresponds roughly to 8 kW of heating power.

This example solves for the temperature only in the susceptor and the quartz tube. It models heat flux between the susceptor and the tube with surface-to-surface radiation. The only part where heating occurs is in the inner part of the susceptor, which is made of pure graphite. The graphite foam outside has a lower thermal conductivity and zero electric conductivity. The bottom surface of the hole in the susceptor is modeled as a highly conductive layer because it is made of silicon carbide, which has a very high thermal conductivity. The layer's thickness is 1 mm, which also includes the wafer thickness.

The following table summarizes all the material properties used in this simulation:

PARAMETER	COPPER	GRAPHITE	GRAPHITE FELT	QUARTZ	SILICON CARBIDE
Heat capacity	584	710	200	820	1200
Density	8700	1950	120	2600	3200
Thermal conductivity	400	150 · (300/T)	0.3	3	450 · (300/T)^0.75
Electric conductivity	5.998e7	1e3	100	1e-12	1e3
Permittivity	1	1	1	4.2	10
Emissivity	0.5	1	1	0.7	0.5

The temperature dependence of the thermal conductivity is necessary due to the high temperature range in the susceptor.

In this model you neglect the temperature dependence of the electric conductivity. This enables COMSOL Multiphysics to solve the application modes sequentially, which saves memory. The problem can be solved fully coupled with a temperature dependent conductivity on high-performance computers. By using the Induction Heating predefined multiphysics coupling, it is easy to take the electro-thermal interaction into account.

Results and Discussions

The temperature difference over the wafer should be as small as possible; a large difference can cause nonuniform thicknesses and properties in the grown layer. The heating cycle is also interesting to view, and Figure 5-14 compares the temperature at the center of the wafer to that at the edge during 1 hour of heating at 8 kW.

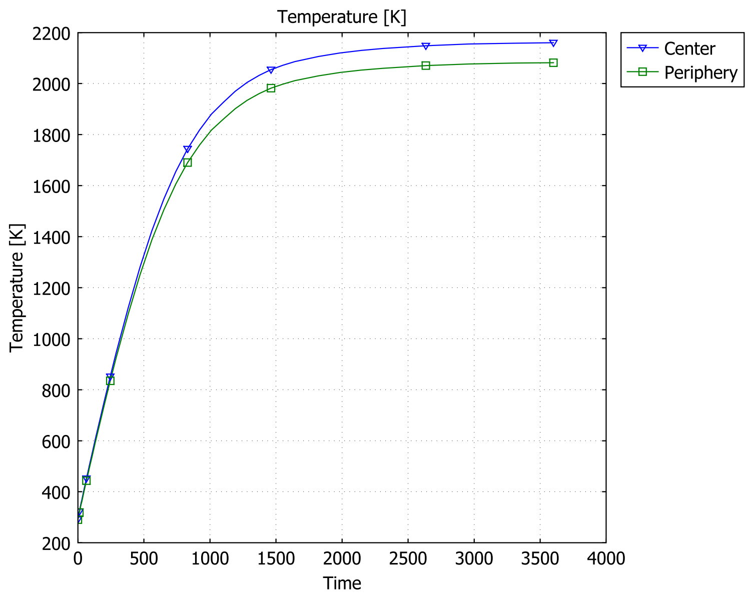


Figure 5-14: The temperature at two locations on the wafer during a heating cycle of one hour.

The graphite felt around the susceptor acts as a high-temperature thermal insulator for the system. This makes the furnace more efficient in terms of power usage, and it also keeps high temperatures away from the quartz tube. The temperature distribution in Figure 5-15 shows how hot the quartz tube gets and how well the graphite felt insulates the susceptor. The maximum temperature of the quartz tube is approximately 900 K, and that tube should be water or air cooled to further reduce its temperature; this model takes only radiation to the ambient into account. COMSOL Multiphysics use extrusion coupling variables to copy the solution to the other symmetry half in this model, producing a more complete plot.

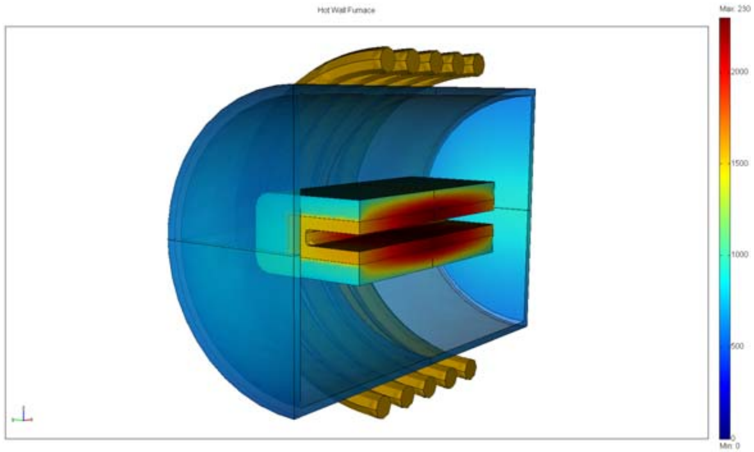


Figure 5-15: Temperature distribution for the important parts of the furnace. The coils are plotted only to show their location, there is no temperature information on their surfaces.

The temperature distribution across the wafer can often be seen in variations of the layer properties, so it is crucial that the temperature be uniform across the surface. The surface plot in Figure 5-16 shows the temperature over one half of the silicon-carbide wafer. The 150 °C difference is rather large. Gas flow is another important source of nonuniformities, but this simulation does not consider it.

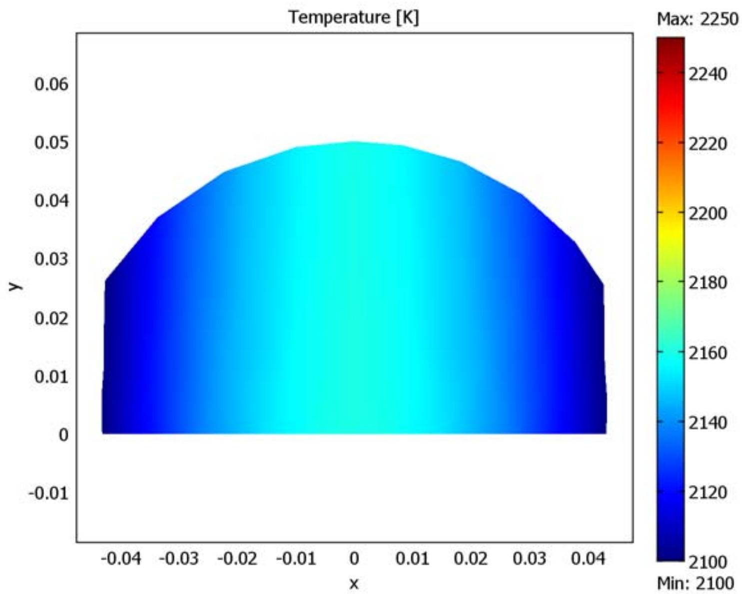


Figure 5-16: Temperature distribution across the surface of the wafer.

Model Library path: `ACDC_Module/General_Industrial_Applications/hot_wall_furnace`

Note: This model requires both the AC/DC Module and the Heat Transfer Module.

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator**, select **3D** in the **Space dimension** list.

2 Select **AC/DC Module>Electro-Thermal Interaction>Induction Heating>Transient analysis**.

3 Select **Vector, Lagrange - Linear** from the **Element** list.

4 Click **OK** to close the **Model Navigator** dialog box.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Constants**.

2 In the **Constants** dialog box, define the following constant with name and expressions. The description field is optional.

NAME	EXPRESSION	DESCRIPTION
`I_coil`	`2150[A]`	`Coil current`

3 Click **OK**.

4 From the **Options** menu, choose **Expressions>Scalar Expressions**.

5 In the **Scalar Expressions** dialog box, define the following expressions:

NAME	EXPRESSION	DESCRIPTION
`Js_coil`	`I_coil/C_coil`	`Surface current on coil`
`Pav_coil`	`4*2.5*0.5*real(I_coil* conj(V_coil))`	`Heating power in coils`

Because of the use of integration coupling variables (`C_coil` and `V_coil`), COMSOL Multiphysics cannot determine the unit for other variables that directly or indirectly depend on these integration coupling variables. This causes the warnings for inconsistent units here and in the specification of the surface current density. You can disregard these warnings.

6 Click **OK**.

GEOMETRY MODELING

The geometry modeling of this model is rather extensive, so it is possible to import the furnace geometry from a binary file. Select the section that you prefer.

Importing the Geometry from a Binary File

1 From the **File** menu, select **Import>CAD Data From File**.

2 In the **Import CAD Data From File** dialog box, make sure that either **COMSOL Multiphysics file** or **All 3D CAD files** is selected in the **Files of type** list.

3 From the COMSOL Multiphysics installation directory, go to the model library path given on page 273. Select the `hot_wall_furnace.mphbin` file, and click **Import**.

4 Skip the section "Creating the Geometry from Scratch" and continue at "Physics Setting" on page 278.

Creating the Geometry from Scratch

1 From the **Draw** menu, choose **Work-Plane Settings**, and click the **y-z** option button.

2 Click **OK**.

3 Create three rectangles with the specifications in the table below. You create them easiest from the dialog box that you open by choosing **Draw>Specify Objects>Rectangle**.

NAME	WIDTH	HEIGHT	X	Y	BASE
R1	`0.075`	`0.02`	`0`	`-0.01`	Corner
R2	`0.1`	`0.05`	`0`	`-0.025`	Corner
R3	`0.15`	`0.1`	`0`	`-0.05`	Corner
R4	`0.1`	`0.05`	`0`	`-0.025`	Corner

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

5 Choose **Fillet/Chamfer** from the **Draw** menu.

6 In the **Fillet/Chamfer** dialog box, expand rectangle R1, and select its vertex numbers 2 and 3. Enter `5e-3` in the **Radius** edit field.

7 Click **Apply**. The former rectangle R1 is now labeled CO1.

8 Then expand rectangle R3, select its Vertices 2 and 3, and enter `0.02` in the **Radius** edit field.

9 Click **OK**. The former rectangle R3 is now labeled CO2.

10 Select the objects labeled R2 and CO1, and click the **Difference** button on the Draw toolbar.

11 Select the objects labeled R4 and CO2, and click the **Difference** button again.

12 From the **Draw** menu, choose **Extrude**.

13 In the **Extrude** dialog box, select the objects CO1 and CO3, and enter `0.15` in the **Distance** edit field.

14 Click **OK**. The program switches to the 3D view, where the extruded objects represents the susceptor and its thermal insulation.

15 From the **Draw** menu, choose **Work-Plane Settings** again, and click the **z-x** option button to use a different work plane.

16 Click **OK**.

17 Press Ctrl+A to select all objects in the work plane, and then press the **Delete** key to clear the work plane.

18 Create three rectangles with the specifications according to the table below.

NAME	WIDTH	HEIGHT	X	Y	BASE
R1	0.18	0.01	0	0.25	Corner
R2	0.01	0.25	0.17	0	Corner
R3	0.45	0.5	0	0	Corner

19 Select both rectangles and click the **Union** toolbar button.

20 Click the **Delete Interior Boundaries** toolbar button to remove unnecessary boundaries.

21 Use the menu option **Draw>Specify Objects>Circle** to create three circles with the specifications according to the table below.

NAME	RADIUS	X	Y	BASE
C1	0.015	0.21	0	Center
C2	0.015	0.21	5e-2	Center
C3	0.015	0.21	1e-1	Center

22 Create a line going from the coordinate (0.195, 0) to the coordinate (0.225,0).

23 Select the line B1 and the circle C1 and click the **Coerce to Solid** toolbar button.

24 Click the **Split Object** toolbar button and select the side of the circle labeled CO4.

25 Press the **Delete** key to delete this side.

26 From the **Draw** menu, choose **Revolve**.

27 Select the objects CO2, CO3, C2, C3, R3. Enter `-180` in the **α2** edit field.

28 Click **OK**. You are in the 3D view again, and now you have created the quartz tube and the coils. One of the coils was split in half due to the use of symmetry in this model. Define one last work plane to draw the items inside the susceptor.

29 From the **Draw** menu, choose **Work-Plane Settings** again, and click the **x-y** option button.

30 Click **OK**.

31 Press Ctrl+A to select all objects in the work plane, and then press the **Delete** key to clear the work plane.

32 Create a circles with radius 0.05 centered at the origin.

33 Create a line going from the coordinate (0, 0) to the coordinate (0.05,0).

34 Select the circle and the line, click the **Coerce to Solid** toolbar button, and then click the **Split Object** toolbar button.

35 Select and delete the lower part of the circle labeled CO2.

36 Create a rectangle with width 0.15, height 0.055, and the corner coordinate in the origin.

37 Choose **Embed** from the **Draw** menu, select all objects (CO2 and R1), and click **OK**.

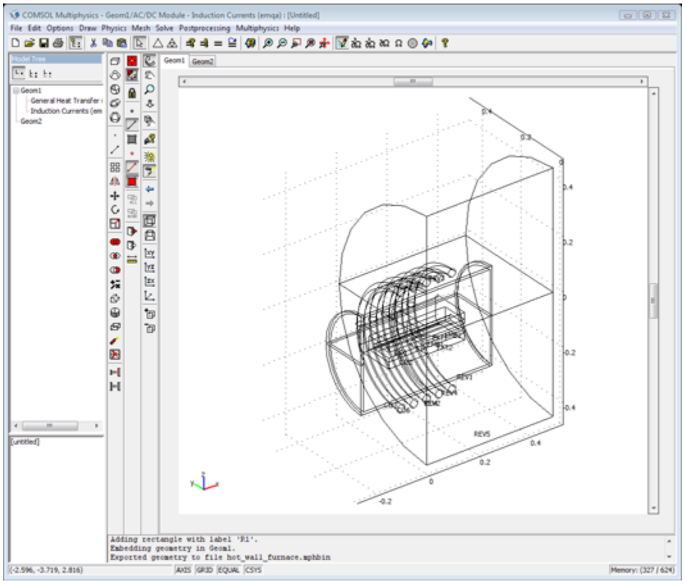
The simulation domain is now finished. However, to improve the plot view, some parts of the furnace can be mirrored to the other side of the $x = 0$ symmetry plane. The mirrored part is only used for postprocessing and does not influence the simulation.

1 Select all objects except the largest cylinder. The selected objects should have the labels EXT1, EXT2, REV1, REV2, REV3, REV4, EMB1, and EMB2.

2 Click the **Mirror** toolbar button.

3 In the **Mirror** dialog box, set the **Normal vector** edit fields to (1,0,0), and click **OK**.

4 Click **Zoom Extents** and click the **Headlight** toolbar button. With the proper objects selected the geometry should look like that in the figure below.



PHYSICS SETTING

Variables

1 From the **Physics** menu, choose **Scalar Variables**.

2 Enter `20e3` in the **nu_emqa** edit field, then click **OK**.

3 From the **Options** menu, choose **Integration Coupling Variables>Edge Variables**.

4 In the **Edge Integration Variables** dialog box, enter the following variables with names and expressions at the specified edges. The **Global destination** check box should be selected for all variables.

NAME	EDGES	EXPRESSION	DESCRIPTION
`C_coil`	221, 222, 230, 232	1	Gives the circumference of one coil
`V_coil`	217	`Ex_emqa*t1x+ Ey_emqa*t1y+ Ez_emqa*t1z`	Induced voltage along one coil
	220	`-(Ex_emqa*t1x+ Ey_emqa*t1y+ Ez_emqa*t1z)`	Induced voltage along one coil

5 From the **Options** menu, choose **Extrusion Coupling Variables>Subdomain Variables**.

6 In the **Subdomain Extrusion Variables** dialog box, select Subdomains 9, 11, and 12, and enter `T_map` in the **Name** column.

7 Then click the **General Transformation** option button. Leave the transformation expressions at their defaults.

8 Enter the `T` in the **Expression** column in the same row as the `T_map` variable.

9 Click the **Destination** tab. Make sure that the **Geometry** list has **Geom1** chosen, then choose **Subdomain** in the **Level** list.

10 Select Subdomains 1, 2, and 3. Select the **Use the selected subdomains as destination** check box.

11 Enter `-x` in the **x** edit field for the **Destination transform**.

12 Click **OK**. The temperature in Subdomains 9, 11, and 12 are now mirrored into Subdomains 1, 2, and 3. The mirrored version of the temperature is available there as the variable `T_map`.

13 Choose **Expressions>Boundary Expressions** from the **Options** menu.

14 In the **Boundary Expressions** dialog box, define a variable named `T_bnd` with the following expressions for the specified boundary numbers. Due to the large number of boundaries, there is a technique is outlined below the table that selects all these boundaries from the geometry directly.

NAME	EXPRESSION FOR BOUNDARIES 68–94, 97, 104–112, 133–140	EXPRESSION FOR BOUNDARIES 1–32, 51
`T_bnd`	`T`	`T_map`

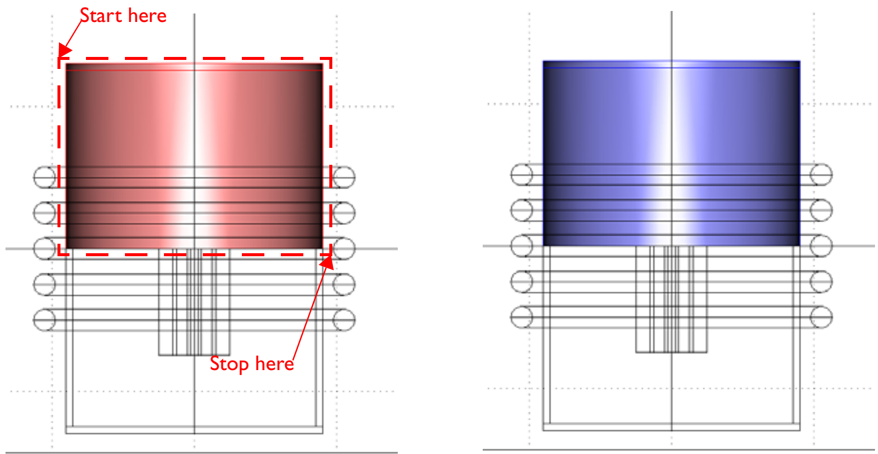
15 Click **OK**.

16 The technique involves rectangular mouse selections in 3D, just skip Steps 16–25 if you prefer to select the boundaries by number according to the table above. By default the mouse controls the 3D view (panning, rotating, zooming), so first click the **Orbit/Pan/Zoom** toolbar button to disable the 3D view control.

17 Then click the **Go to ZX View** toolbar button to get a proper view for the selection.

18 Place the mouse pointer just outside the quartz tube and click and drag it to the other side of the tube that you solve for. View the left figure below for guidance,

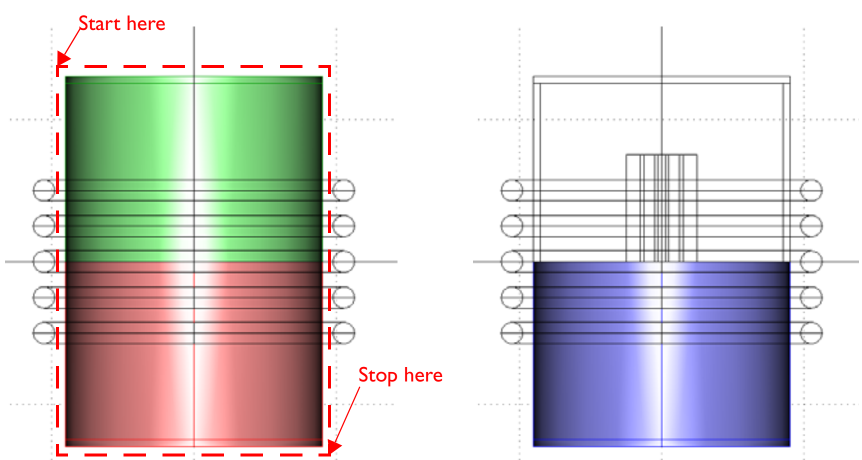
which shows the selection rectangle to draw with the mouse and the selection after you release the mouse button.



19 Right-click with the mouse on the selected region to lock the selection. The selected objects then get a blue color like the right figure above.

20 Go back to the **Boundary Expressions** dialog box and enter `T` in the **Expression** column in the same row as the `T_bnd` name.

21 Do a new selection starting at the same point as before, but extend it to include the entire tube. The selection now looks like that in the left figure below.



22 The green selection shows the locked boundaries (blue before) that are scheduled for deselection. The pink ones are the boundaries you selected that where not selected before. Right-click on the selection to confirm the green deselection and lock the pink selection. The view should now look like the right figure above.

23 Go to the **Boundary Expressions** dialog box and enter `T_map` in the **Expression** column in the same row as the `T_bnd` name.

24 Click **OK** to close the dialog box.

25 Click the **Go to Default 3D View** toolbar button and then click the **Orbit/Pan/Zoom** toolbar button to restore the settings as it were before.

PHYSICS SETTINGS

Subdomain Settings—General Heat Transfer

For multiphysics modeling with several application modes present in the model, there is a convenient feature on the left side of the graphical user interface. It is called the Model Tree, and you can show it or hide it by clicking the **Model Tree** toolbar button. Adjust the size of the Model Tree area and the size of its frames so it looks similar to what you see in the figure of the final geometry above. The steps below assume that you have the **Model Tree** visible and sized correctly. Also, make sure to select the **Detail** view.

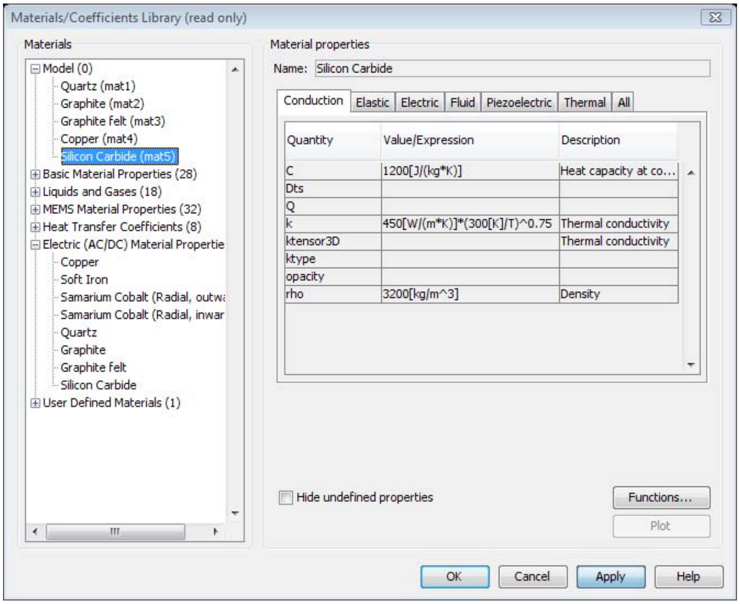
1 In the **Model Tree**, double click on the item **Geom1>General Heat Transfer (htgh)> Subdomain Settings**.

2 In the **Subdomain Settings** dialog box, select Subdomain 9 and then click the **Load** button to open the **Materials/Coefficients Library** dialog box.

3 Select the **Quartz** material under the `acdc_lib.txt` library.

4 Click **Apply**. This adds the material **Quartz** to the model. You see which materials you have added to the model when you expand the **Model** node in the **Materials** list to the left in the **Materials/Coefficients Library** dialog box.

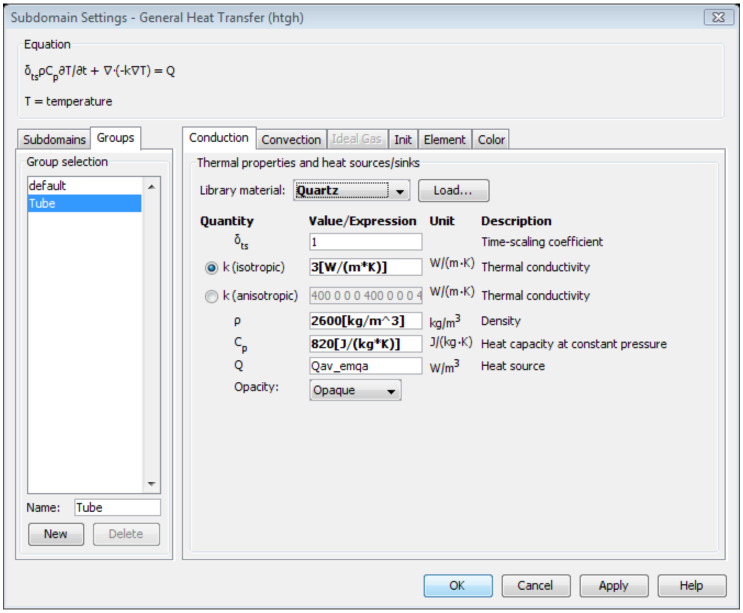
5 Repeat the two last steps for the materials **Graphite**, **Graphite felt**, **Copper**, and **Silicon Carbide**. Before you close the dialog box, make sure that you see all five materials under **Model** in the **Materials** tree view.



6 Click **Cancel** to close the **Materials/Coefficients Library** dialog box. The selected materials are now available in the **Library material** list.

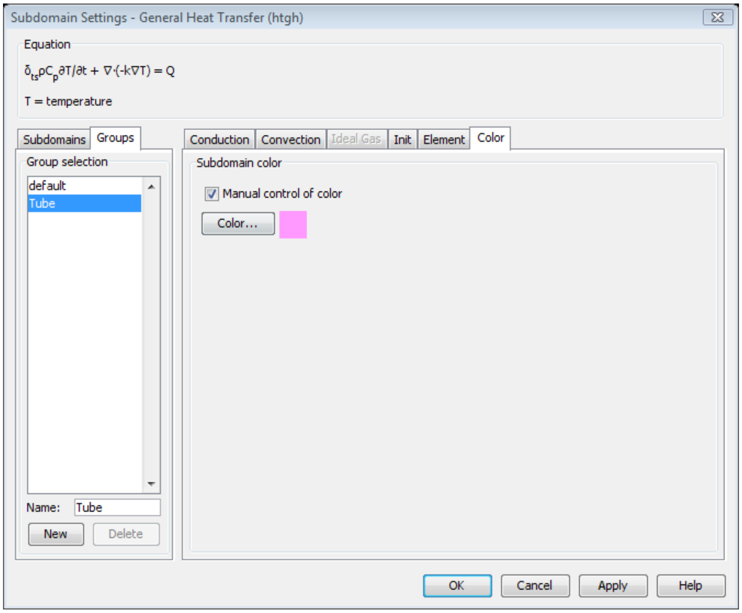
7 Choose **Quartz** from the **Library material** list.

8 Click the **Groups** tab, and enter Tube in the **Name** edit field.



9 Click the **Color** tab, select the **Manual control of color** check box, and click the **Color** button.

10 Select the a bright purple color from the palette, or enter (255, 153, 253) under the **RGB** tab. Click **OK**.



These last two steps and future steps that change the color of a group are only necessary to get a nicer view of the geometry, so you can skip these steps.

11 In the **Subdomain Settings** dialog box, define settings for the other subdomains according to the table below. If desired, repeat Steps 9 and 10 to define a color for each group.

SETTINGS	SUBDOMAIN 11	SUBDOMAIN 12
Material	Graphite felt	Graphite
Group name	Felt	Susceptor
Color	(153, 153, 153)	(51, 51, 51)

12 Under the **Groups** tab, select the group **default** in the **Group selection** field. Click the **Subdomains** tab, and clear the **Active in this subdomain** check box.

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

Subdomain Settings—Induction Currents

1 In the **Model Tree**, double click the item **Geom1>Induction Currents (emqa)> Subdomain Settings**.

2 Define settings for the other subdomains according to the table below in the same as for the Heat Transfer application mode. If desired, repeat Steps 9 and 10 to define a color for each group.

SETTINGS	SUBDOMAIN 9	SUBDOMAIN 11	SUBDOMAIN 12
Material	Quartz	Graphite felt	Graphite
Group name	`Tube`	`Felt`	`Susceptor`

3 Click the **Subdomains** tab and select Subdomains 1, 2, 3, 4, 5, 6, 8, 13, and 14. Clear the **Active in this subdomain** check box.

4 Select the remaining Subdomains 7 and 10, click the **Groups** tab, and type `Air` in the **Name** edit field.

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

The coils are not included in this simulation since the skin-depth is very short for the applied frequency. The coils are instead modeled as a surface current boundary condition.

Boundary Conditions—General Heat Transfer

1 In the **Model Tree**, double click on the item **Geom1>General Heat Transfer (htgh)> Boundary Settings**.

2 In the **Boundary Settings** dialog box, define settings according to the table below.

SETTINGS	BOUNDARIES 76, 86, 91, 105–108, 110–112, 133, 134	BOUNDARIES 81, 104	BOUNDARIES 73, 94, 136, 137	BOUNDARIES 70, 97, 135, 138–140
Material	Graphite felt	Silicon Carbide	Quartz	Quartz
Boundary condition	Heat flux	Heat flux	Heat flux	Heat flux
Radiation type	Surface-to-Surface	Surface-to-Surface	Surface-to-Surface	Surface-to-Ambient
Tamb	`T`	`T`	`T`	`293.15`
Group name	`Susceptor`	`Slide`	`Tube_inner`	`Tube_outer`

3 Select Boundaries 81 and 104 again, click the **Highly Conductive Layer** tab, and select the **Enable heat transfer in highly conductive layer** check box.

4 In the **Distance** edit field type `1e-3` to set the thickness of this highly conductive layer.

5 Click **OK** to close the **Boundary Settings** dialog box.

Boundary Conditions—Induction Currents

1 In the **Model Tree** double click the item **Geom1>Induction Currents (emqa)>Boundary Settings**.

2 In the **Boundary Settings** dialog box define settings according to the table below.

SETTINGS	BOUNDARIES 60, 66, 68, 71, 74, 77, 80, 82, 84, 87, 90, 92, 95, 101	BOUNDARIES 65, 67, 100, 102, 113, 114, 116, 117, 119, 120–124, 126, 127, 129-132	ALL OTHERS
Boundary condition	Electric insulation	Surface current	Magnetic insulation
$\mathbf{J}_s$		`0`	
		`Js_coil*t2y`	
		`Js_coil*t2z`	
Group name	`Symmetry`	`Coil`	

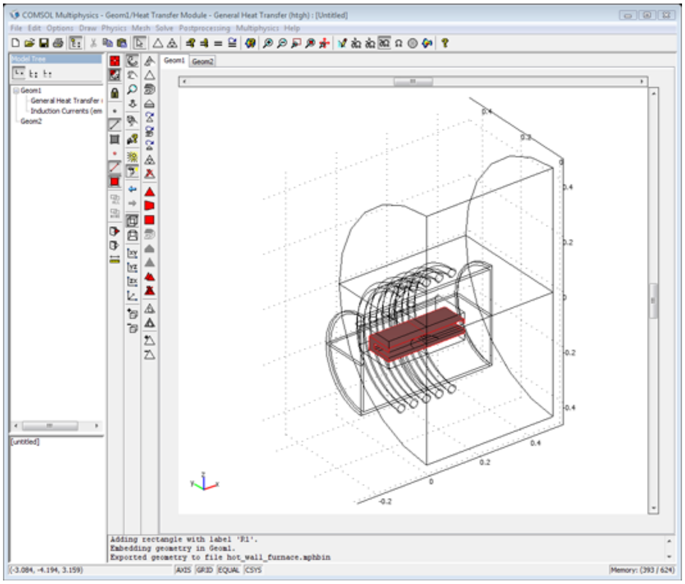
3 Click **OK** to close the **Boundary Settings** dialog box.

MESH PARAMETERS

1 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.

2 Click the **Custom mesh size** option button, and enter `0.5` in the **Mesh curvature factor** edit field.

3 Click the **Boundary** tab, and select all boundaries between the felt and the susceptor. The numbers of these boundaries are 16, 18, 23, 29, 79, 89, 109, and 134.



4 Enter `6e-3` in the **Maximum element size** edit field.

5 Click **Remesh**.

6 Click **OK**.

SOLVER PARAMETERS

1 Open the **Solver Parameters** dialog box from the **Solve** menu.

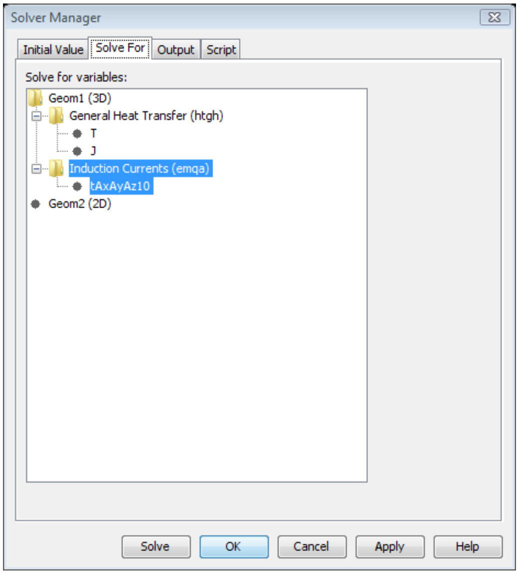
2 With the **Auto select solver** check box selected, select **Stationary** from the **Analysis** list.

3 Click the **Stationary** tab and choose **Linear** from the **Linearity** list (the model is linear, but the automatic selection chooses the nonlinear solver for this model because of the coupling variables).

4 Click **OK**.

5 Open the **Solver Manager** dialog box from the **Solve** menu.

6 Click the **Solve For** tab, and select only the **Induction Currents (emqa)** application mode in the tree view.



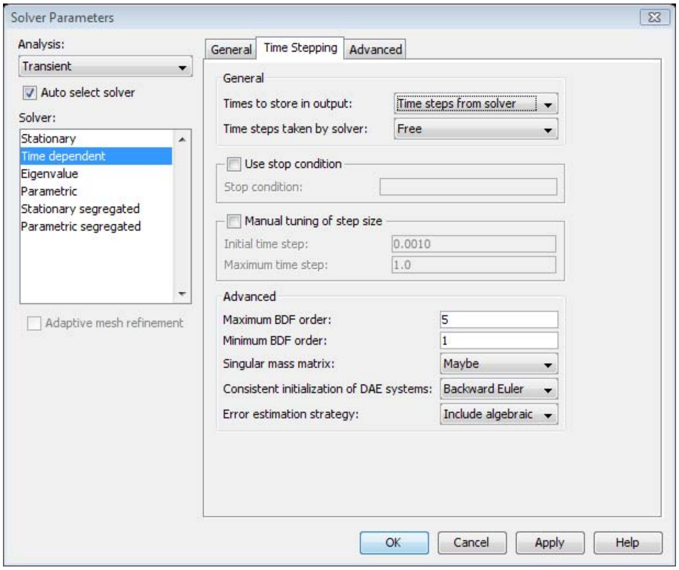
7 Click **OK**.

8 Click the **Solve** toolbar button.

9 When the solution process is done open the **Solver Parameters** dialog box again.

10 From the **Analysis** list, select **Transient**, then type `0 3600` in the **Times** edit field.

11 Click the **Time Stepping** tab and choose **Time steps from solver** in the **Times to store in output** list.



12 Click **OK**.

13 Open the **Solver Manager**, select to solve only for the **General Heat Transfer (htgh)** application mode on the **Solve For** page.

14 Click the **Initial value** tab, and in the frame **Values of variables not solved for and linearization point** click the **Current solution** button. This ensures that the solution for the induction currents problem is kept while solving the transient heat transfer problem.

15 Click **OK** to close the **Solver Manager**.

16 Click the **Solve** toolbar button.

POSTPROCESSING AND VISUALIZATION

1 Open the **Plot Parameters** dialog box from the **Postprocessing** menu.

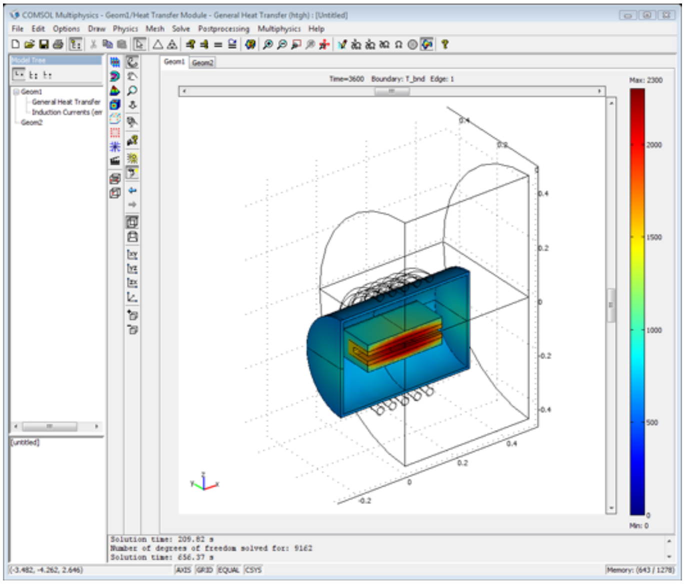
2 Under the **General** tab, clear the **Slice** and **Geometry edges** check boxes, and select the **Boundary** and **Edge** check boxes.

3 Click the **Boundary** tab and enter `T_bnd` in the **Expression** edit field.

4 Click the **Range** button. In the dialog box that appears, clear the **Auto** check box, and enter `0` and `2300` in the **Min** and **Max** edit field.

5 Click the **Edge** tab, enter 1 in the **Expression** edit field, click the **Uniform** color option button, and select black color for the edges.

6 Click **OK** to show the plot in the main window.



To get the plot in Figure 5-15 on page 272 you need some extra steps. It is necessary to plot into a separate figure and to do overlaid plots. To make plots only in certain regions you either use **Suppress>Boundaries** from the **Options** menu, or define boundary expressions of the quantity you whish to plot. These techniques makes it possible to remove boundaries from the plot. Then you do separate plots for the susceptor, the tube, and the coils. Remember to select the **Keep current plot** check box in the **Plot Parameters** dialog. Make the plot for the coils with a uniform copper-like color. In the created figure you can click the **Edit Plot** toolbar button, and select individual properties for the plots you made. In Figure 5-15 on page 272 there is also a transparency set on the tube plot.

The next step is to generate line plots of the temperature versus time at two location on the silicon carbide wafer, center and periphery.

1 From the **Postprocessing** menu, choose **Domain Plot Parameters**.

2 In the **Domain Plot Parameters** dialog box, click the **Point** tab and select points number 64 and 102. This is the center and periphery of the SiC wafer.

3 Click the **Line settings** button, and choose **Cycle** in the **Line marker** list.

4 Select the **Legend** check box and click **OK**.
5 Click **OK** again to get the line plot in a new figure window.
6 In the new figure, click the **Edit Plot** toolbar button. Select the **Line (64)** item from the tree view, and then type `Center Periphery` in the **Legends** edit field.
7 Click **OK**. You should now see a plot similar to that in Figure 5-14 on page 271.

The final plot is a surface plot of the temperature across SiC wafer. Because half the wafer is outside the simulated region you must use the boundary variable, `T_bnd`, to see the temperature.

1 Open **Domain Plot Parameters**, click the **General** tab and select only the solution at time **3600** from the list of time steps.
2 Click the **Surface** tab, select Boundaries 51 and 81, enter `T_bnd` in the **Expression** edit field, and choose **xy-plane** from the list in **x- and y-axis data**.
3 Click the **Range** button. In the dialog box that appears, enter `2100` and `2300` in the **Min** and **Max** edit fields, respectively.
4 Click **OK** twice to get the surface plot in the opened figure window.
5 In the figure, click the **Edit Plot** toolbar button. Select the **Axis** item from the tree view, and then enter `Temperature [K]` in the **Title** edit field.
6 Click **OK** to get the plot in Figure 5-16 on page 273.

An RFID System

This example illustrates the modeling of a reader-transponder pair for radio-frequency identification (RFID) applications.

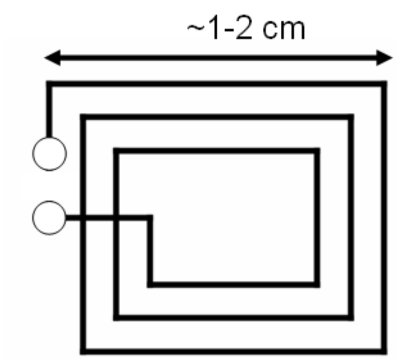
Introduction

An RFID system consists of two main parts:

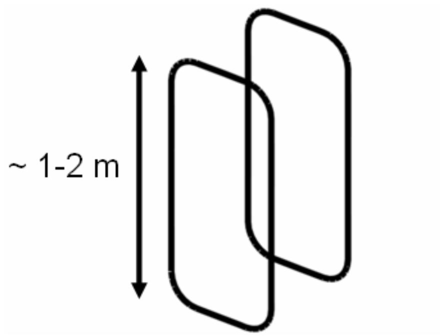
- A tag or transponder with a printed circuit-board (PCB) antenna
- A reader unit with a larger RF antenna

The reader antenna generates an electromagnetic field that energizes a chip (IC-circuit) inside the tag. The electromagnetic field is modulated by the tag's circuit and the modulated signal is recovered by the reader antenna.

The transponder antenna is typically a conductor pattern on a substrate:



and a common type of reader antenna is a larger dual coil:



INDUCTIVE COUPLING

The coupling of the antennas is mainly inductive and is characterized by the mutual inductance, denoted L_{12}. The mutual inductance is defined as the total magnetic flux intercepted by one antenna for a unit current flowing in the other antenna

$$L_{12} = \frac{\iint_{S_2} \mathbf{B} \cdot \mathbf{n} dS}{I_1}$$

where S_2 is the area of coil number 2, $\mathbf{B}$ is the flux intercepted by coil 2, I_1 is the current running in coil number 1, $\mathbf{n} = (n_x, n_y, n_z)$ is the unit surface normal vector, and dS is an infinitesimal area element.

It is possible to transform this into a simpler line integral by using the magnetic vector potential:

$$\mathbf{B} = \nabla \times \mathbf{A}$$

together with the so-called Stokes theorem, stating that a surface integral of the curl of a field equals the closed line integral over the rim of the surface:

$$L_{12} = \frac{\iint_{S_2} (\nabla \times \mathbf{A}) \cdot \mathbf{n} dS}{I_1} = \frac{\oint_{\Gamma_S} \mathbf{A} \cdot \mathbf{t} dl}{I_1}$$

where $\mathbf{t}=(t_x, t_y, t_z)$ is the unit tangent vector of the curve Γ_S, and where dl is an infinitesimal line element.

Because the coupling is dominated by near-field inductive effects, it is sufficient to compute the mutual inductance for the static case (frequency equals zero) and neglect capacitive effects along with wave propagation phenomena. The appropriate application mode to use is called 3D Magnetostatics and is available in the AC/DC Module. It has the magnetic vector potential $\mathbf{A} = (A_x, A_y, A_z)$ as its unknown field quantity.

EDGE COMPUTATIONS

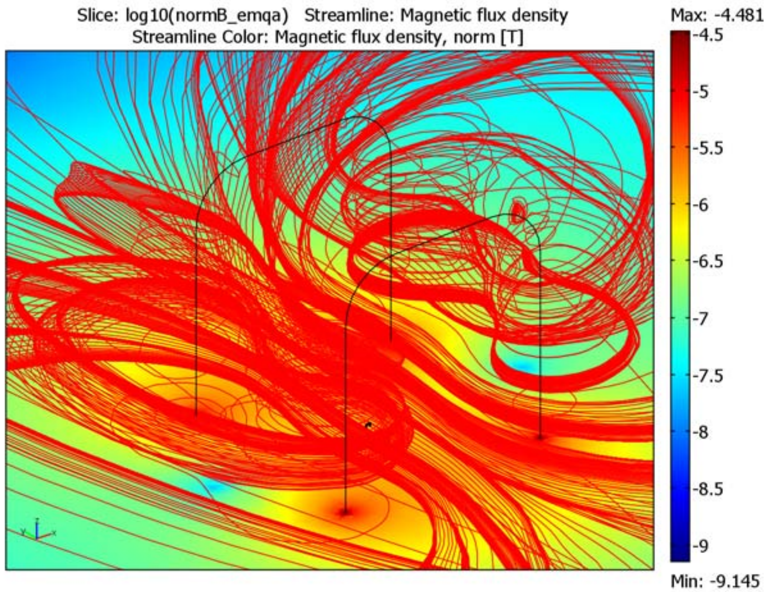
This example approximates coils and wires as edges (1-D entities) embedded in 3D space and the variables and techniques used are somewhat different than for a regular magnetostatics boundary value problem. The magnetic vector potential components are available at edges as the variables `tAx`, `tAy`, and `tAz`, which form the projection of the vector potential on edges. On the other hand, the bulk components are denoted `Ax`, `Ay`, and `Az`.

In COMSOL Multiphysics, each edge has a direction associated with it, and in order to form a closed line integral it is necessary to manually find the direction of each edge and flip this if needed. Flip the edge directions by the following procedure: for each edge that participates in a line integral or line source, define an **Edge Expression** `esgn` that takes the values `-1` and `+1` depending on if the built-in direction of the edge agrees with the global direction of the closed line integral. (It is up to the user to decide for a global direction of a closed line integral.)

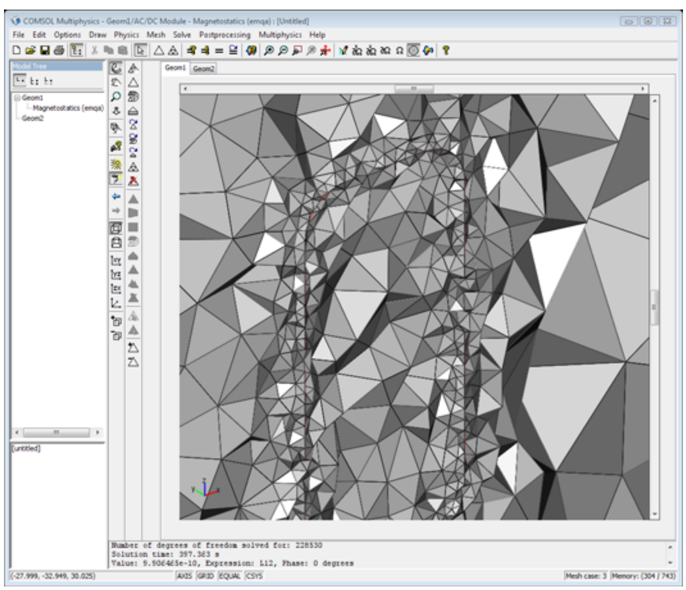
Results

The computed value for the mutual inductance L_{12} is about 1 nH.

The following visualization shows magnetic flux lines and the base-10 logarithm of the magnetic flux intensity as color.



The following visualization shows the edges of the finite elements used:



Model Library path: `ACDC_Module/General_Industrial_Applications/rfid`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 From the **Model Navigator**, set the **Space Dimension** to **3D**.

2 In the list of application modes, select **AC/DC Module>Statics>Magnetostatics**.

3 Click **OK** to open the main GUI.

DRAW MODE

We will start by drawing the transponder antenna in a 2D work plane and then embed it into 3D space.

Transponder Antenna

1 From the **Draw** menu, select **Work-Plane Settings**.

2 Click the **z-x** plane button with y = 0.

3 Click **OK** to create the work plane.

4 From the **Draw** menu, select **Specify Objects>Line**.

5 In the **Line** dialog box, type a space separated list of vertex coordinates according to the following:

X: `0.8875 0.8875 0.9125 0.9125 0.89 0.89 0.91 0.91 0.8925 0.8925 0.9075 0.9075 0.895 0.895 0.8875`

Y: `0 0.01 0.01 -0.01 -0.01 0.0075 0.0075 -0.0075 -0.0075 0.005 0.005 -0.005 -0.005 0 0`

6 Click **OK** to create the transponder profile.

7 Click the **Zoom Extents** button on the main toolbar to automatically adjust the axis settings to get a good view of the transponder profile.

Reader Antenna

Next, create the reader antenna.

1 From the **Draw** menu, select **Specify Objects>Rectangle**.

2 Type values for a rectangle with **Width**: `1.8`, **Height**: `0.8`, **Base**: **Corner**, **x:** `0`, **y:** `-0.4`, **Style: Curve**.

3 Click **OK** to create the rectangle.

4 Click the **Zoom Extents** button.

5 To create rounded corners, or fillets, for the rectangle, click the **Fillet/Chamfer** button.

6 In the **Vertex selection** list, click the object **R1** (the rectangle) and select all 4 vertices by using Ctrl+left click or Ctrl+A.

7 Specify a **Radius** of `0.2`.

8 Click **OK** to apply the fillets.

Embedding into 3D Space

Now embed the two flat shapes into 3D space.

1 From the **Draw** menu, select **Embed**.

2 Highlight both objects **B1** and **B2**.

3 Click **OK** to embed the objects into 3D space.

4 Click on the edge on the larger coil to select it. Make sure it is highlighted in red.

5 Click the **Move** button.

6 For the **y-displacement**, type `-0.4`.

7 Click **OK** to displace the large coil element 0.4 m in the negative y direction.

8 To create a mirror-image of the large coil, make sure it is highlighted and click the **Mirror** button.

9 Change the **Normal vector** from `0 0 1` to `0 1 0`.

10 Click **OK** to create the mirrored image.

Truncation Sphere

You also need to truncate the computational domain, because the finite element calculation requires a finite-sized mesh. Do this with a large sphere with center in the middle of the coil system.

1 From the **Draw** menu, select **Sphere**.

2 Set the **Radius** to `3`.

3 Set the **Axis base point** to **x:** `0`, **y:** `0`, **z:** `0.9`.

4 Click **OK** to create the sphere.

Make sure to save the model at this point to be able to revert to this stage if needed.

SUBDOMAIN SETTINGS

1. From the **Physics** menu, select **Subdomain Settings**.
2. The material parameter that you can alter for a magnetostatic simulation is the magnetic permeability μ_r. However, assume that the truncation sphere is filled with air and leave the default setting μ_r=1.
3. Click **OK** to close the dialog box.

BOUNDARY SETTINGS

1. From the **Physics** menu, select **Boundary Settings**.
2. The default boundary condition for the faces that constitute the sphere is **Magnetic insulation**. This allows for surface currents to run across the sphere in such a way as to make the tangential component of the magnetic vector potential equal to zero. This is a good approximation in place of an infinite domain. It approximates that the tangential component of the vector potential approaches zero as the distance from the coils approaches infinity. Thus there is no need to change the boundary settings.
3. Click **OK** to close the dialog box.

EDGE SETTINGS

Now define the driving line currents of the reader antenna. To do this you need to find the direction of all edges of the reader antenna and flip the direction when needed. You also need the suppress edges feature available from the **Options** menu to conveniently visualize and click the edges. This is a bit technical compared to regular use of the graphical user interface, and the reason is that edges fill up a much smaller part of the screen than boundaries and subdomains and hence edges are harder to select by mouse clicking.

To enhance the visualization of edges before you start the selection process, widen the appearance of the edges and turn on a smoothing feature called antialiasing. Hardware acceleration standards supported by many graphics cards support this edge-smoothing feature.

1. From the **Options** menu, select **Preferences**.
2. Click the **Visualization** tab and change the line width from `1.0` to `1.5`.
3. Select the check box for **Antialiasing**.
4. Click **OK** to close the dialog box.

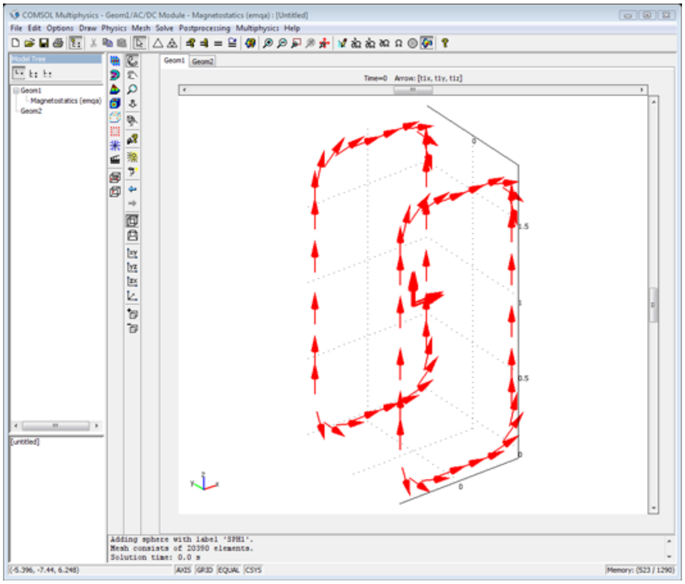
Now, to the edge suppression:

1. From the **Options** menu, select **Suppress>Suppress Edges**. Keep this dialog open but move it aside while performing steps 2 and 3.
2. Click the **Orbit/Pan/Zoom** toolbar button on the Camera toolbar to disable this feature.
3. Click and drag, using the left mouse button, a rubber-band box enclosing the reader antenna and the transponder antenna. Make sure the edges are highlighted in red.
4. In the **Suppress Edges** dialog box, click **Apply**.
5. Click **Invert Suppression**.
6. Click **OK** to close the dialog box.
7. Click the **Zoom Extents** button.
8. Click the **Orbit/Pan/Zoom** toolbar button on the Camera toolbar to enable this feature.

Now visualize the direction of the edges. To do this you need to enter Postprocessing mode. You normally cannot do this until you have solved the problem. However, for these situations there is a feature to help called Update Model.

1. From the **Solve** menu, select **Update Model**. This creates a mesh and puts you in Postprocessing mode. Please note that for a general situation you might need to adjust the mesh parameter settings before selecting **Update Model**.
2. From the **Postprocessing** menu, select **Plot Parameters**.
3. On the **General** page, clear the **Slice** and **Geometry Edges** check boxes
4. Select the **Plot type** called **Arrow**.
5. Click the **Arrow** tab.
6. Change the **Plot arrows on** setting to **Edges**.
7. For the **x component**, **y component**, and **z component** type `t1x`, `t1y`, and `t1z`, respectively. These are the components of the edge tangent vector.
8. For the **Arrow type**, change to **3D arrow**.

9 Click **OK**.



To see which edge correspond to which edge number, we need to turn on the edge label visualization.

1. From the **Options** menu, select **Labels>Show Edge Labels**.
2. To see the edge labels you need to switch to the Edge mode. To alter between the Postprocessing mode and Edge mode, click the corresponding buttons on the main toolbar.

Apply a right-hand rule for the edge directions so that edges circling in the direction from the z-axis to the x-axis with the thumb pointing in the y direction are considered to be of positive sign.

We will now define an **Edge Expression** `esgn` that controls the flip of the edge directions.

1. From the **Options** menu, select **Expressions>Edge Expressions**.

   Use the left mouse button to click the edges or the edge labels and to highlight them in red. Then click the right mouse button to lock the selection. You can undo all selections by pressing Ctrl+D or by selecting **Deselect all** from the **Edit** menu.
2. Select Edges 5, 7, 8, 10, 12, 14, 42, and 44. While all these are highlighted, type `esgn` for the **Name** and `1` for the **Expression**.

3 Select Edges 6, 9, 11, 13, 41, 43, 45, and 46. While all these are highlighted, type `-1` for the **Expression**. These are the edges for which you will flip the direction.

4 Click **OK** to close the dialog box.

5 From the **Physics** menu, select **Edge Settings**.

6 Select Edges 5 through 14 and 41 through 46.

7 In the text field for the **Current in edge segment direction**, type `esgn`.

8 Click **OK** to close the dialog box.

MESH SETTINGS

1 From the **Mesh** menu, select **Free Mesh Parameters**.

2 Change the **Predefined mesh sizes** to **Extremely coarse**.

3 Click the **Remesh** button.

4 Click **OK** to close the dialog box.

SOLVER SETTINGS

1 From the **Solve** menu, select **Solver Parameters**.

2 Click the **Settings** button in the **Linear system solver** area.

3 In the **Linear Systems Solver Settings** dialog box, select **Preconditioner** from the tree view.

4 From the **Preconditioner** list, select **SSOR gauge**.

5 Click **OK** twice to close both dialog boxes.

6 To solve, click the **Solve** button on the Main toolbar.

POSTPROCESSING

Visualization

1 From the **Postprocessing** menu, select **Plot Parameters**.

2 On the **General** page, clear the **Arrow** check box.

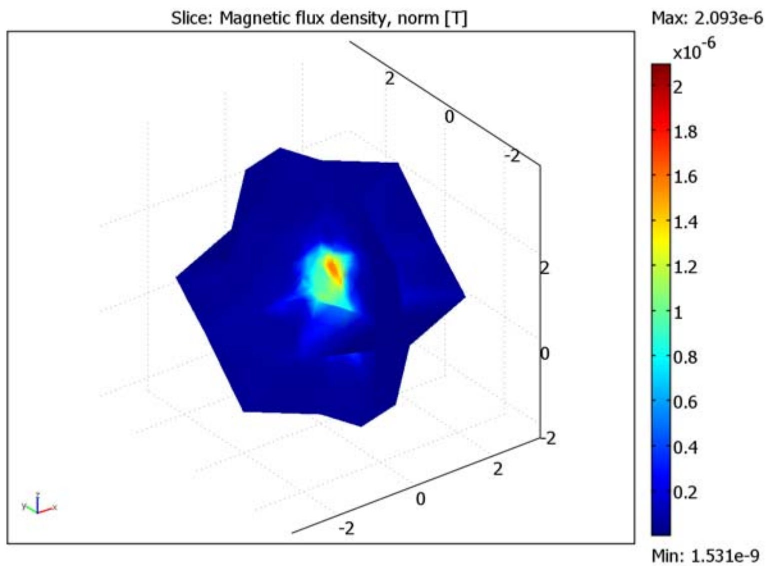
3 Select the **Plot type** called **Slice**.

4 Click the **Slice** tab.

5 Change the **x**, **y**, and **z levels** to `1`, `1`, and `1`, respectively.

6 Click **Apply.**

7 Click **Zoom Extents**.



8 Click the **Streamline tab**.

9 Select the **Streamline plot** check box in the upper left corner.

10 Click **OK** to generate the visualization.

The starting point for the magnetic field lines (streamlines) is randomized and they are unevenly distributed. However, you can see that the magnetic field lines are qualitatively following the expected pattern.

This solution was obtained using a very coarse mesh. Later you will solve for a finer mesh and get better accuracy.

Inductance Computation

Now compute the line integral for the close transponder antenna loop. This also requires that you to flip direction of edges and you will continue to define the `esgn` variable also for the transponder edges. For convenience, first suppress all edges except those of the transponder.

1 From the **Options** menu, select **Suppress>Suppress Edges**.

2 Click the **Orbit/Pan/Zoom** toolbar button on the Camera toolbar to disable this feature.

3 Click the **Zoom Extents** button.

4 Click and drag, using the left mouse button, a rubber-band box enclosing the transponder antenna. Make sure the edges are highlighted in red.

5 In the **Suppress Edges** dialog box, click **Apply**.

6 Click **Invert Suppression**.

7 Click **OK** to close the dialog box.

8 Click the **Zoom Extents** button.

9 Click the **Orbit/Pan/Zoom** toolbar button on the Camera toolbar to enable this feature.

Now visualize the direction of the transponder edges.

1 From the **Postprocessing** menu, select **Plot Parameters**.

2 On the **General** page, clear all check boxes

3 Select **Arrow** and **Edge** for **Plot type**.

4 Click the **Arrow** page.

5 Change the **Plot arrows on** setting to **Edges**.

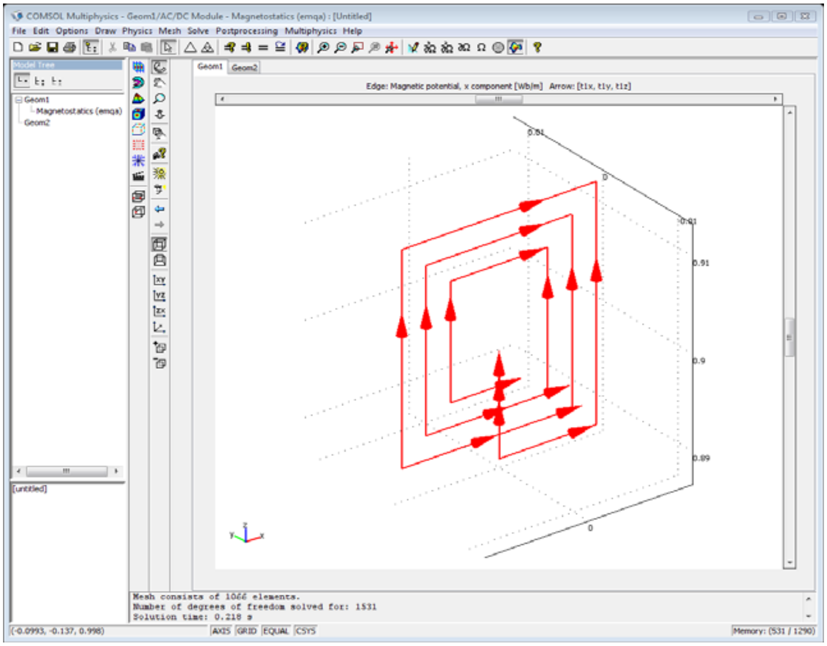
6 For the **x**, **y**, and **z component** make sure they are set to `t1x`, `t1y`, and `t1z`, respectively.

7 For the **Arrow type**, select **3D arrow**, if not already selected.

8 Click the **Edge** page.

9 For the **Edge color**, click **Uniform color**. This aids in the visualization by outlining the edges using a uniform color.

10 Click **OK** to close the dialog box and to create the visualizations.



Repeat the same steps as before to flip the edges. You can toggle between Edge mode and Postprocessing mode to see which edge number has a positive direction.

1. From the **Options** menu, select **Expressions>Edge Expressions**.
2. Use the left mouse button to click the edges or the edge labels and to highlight them in red. Then click the right mouse button to lock the selection. You can undo all selections by clicking Ctrl+D or select **Deselect all** from the **Edit** menu. Alternatively, select directly from the Edge selection list in the dialog box by pressing the Ctrl key while clicking with the left mouse button.
3. Select Edges 15, 17, 18, 20, 21, 23, 29, 31, and 33. While all these are highlighted, type `1` for the **Expression** corresponding to the **Name** `esgn`.
4. Select Edges 16, 19, 22, 30, 32, 34, 38, 39, and 40. While all these are highlighted, type `-1` for the **Expression** corresponding to the **Name** `esgn`.
5. Click **OK** to close the dialog box.

Now compute the integral that gives the mutual inductance:

$$L_{12} = \frac{\oint_{\Gamma_S} \mathbf{A} \cdot \mathbf{t} dl}{I_1} = \oint_{\Gamma_S} \mathbf{A} \cdot \mathbf{t} dl$$

The current of the reader antenna is set to unity, so there is no computation needed for the denominator I_1.

For computing the line integral for the mutual inductance use a so-called edge integration coupling variable that you name `L12` and that sums up the contributions to the closed-line integral for each individual segment.

The integrand at each segment is `esgn*tA`, where `tA` is the projection of the magnetic vector potential in the direction of the unit tangent vector to that edge. The magnetic vector potential components at edges are available as `tAx`, `tAy`, and `tAz` and the edge-tangent vector components are denoted `t1x`, `t1y`, and `t1z`. The tangent vector terminology is inherited from the fact that for a surface the tangent vector comes in pairs as `t1x`, `t1y`, `t1z`, and `t2x`, `t2y`, `t2z`. The projection `tA` is then defined by dot-product of the tangent vector and the magnetic potential: `tA=tAx*t1x+tAy*t1y+tAz*t1z`. Define `tA` as a scalar expression that is available globally.

1 From the **Options** menu, select **Expressions>Scalar Expressions**.
2 Define a **Scalar Expression** with name `tA` and **Expression** `tAx*t1x+tAy*t1y+tAz*t1z.`
3 Click **OK**.

To define the edge integration coupling variable:

1 From the **Options** menu, select **Integration Coupling Variables>Edge Variables**.
2 Select Edges 15–23, 29–34, and 38–40. For this selection you can use Ctrl+left click.
3 Define a variable with the **Name** `L12` and **Expression** `tA*esgn`. You can leave the **Integration order** and **Global destination** by their defaults. The variable `L12` will have global scope for use in postprocessing - it can be used in any of the postprocessing dialog boxes.
4 Click **OK**.

You need to explicitly tell the software to evaluate these newly defined variables:

1 From the **Solve** menu, select **Update Model**. This re-evaluates all variables without re-solving.

To get the value of the mutual inductance variable `L12`:

1 From the **Postprocessing** menu, select **Data Display>Global**.

2 For the **Expression to evaluate**, type `L12`.

Click **OK** to evaluate. A value of about $9 \cdot 10^{-10}$ is displayed in the history log at the bottom of the GUI window. This corresponds to a mutual inductance of 0.9 nH.

SOLVING FOR A FINER MESH USING GMG+GMRES SOLVER

Next, solve the problem using the geometric multigrid (GMG) preconditioner and the iterative FGMRES linear system solver. The GMG preconditioner creates a hierarchy of meshes and solves for the coarsest mesh using a direct solver. Then it interpolates the results to an automatically or manually created finer mesh in a process toward a final solution. In this case, GMG is terminated before it reaches the final solution and instead the partial results is used as a so called preconditioner for the FGMRES iterative solver. This combination of iterative solver and preconditioner is very efficient for solving a large class of problems and proves efficient also for this example.

1 From the **Solve** menu, select **Solver Parameters**.

2 Change the **Linear system solver** to **FGMRES**.

3 Click the **Settings** button.

4 In the **Linear Systems Solver Settings** dialog box, select **Preconditioner** from the tree view.

5 From the **Preconditioner** list, select **Geometric multigrid**.

6 From the **Hierarchy generation method** list, select **Refine mesh**.

7 Click **OK** twice to close both dialog boxes.

8 Click the **Solve** button.

Visualization

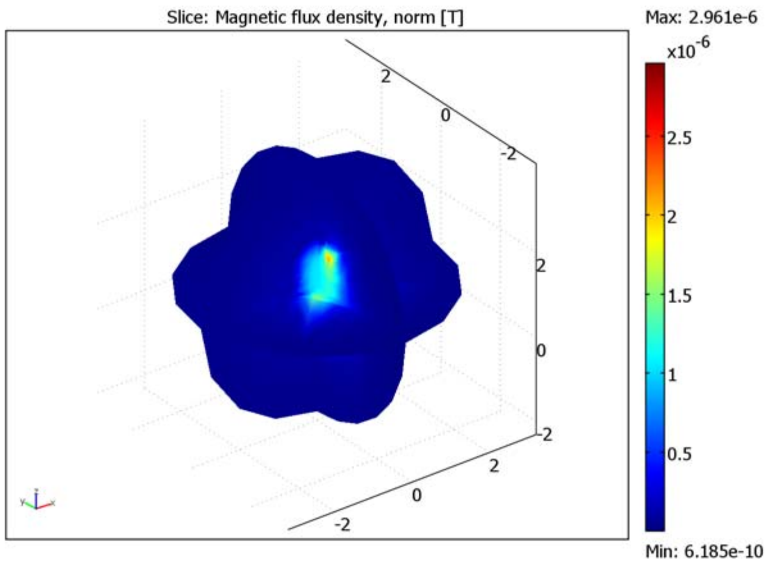
1 From the **Postprocessing** menu, select **Plot Parameters**.

2 On the **General** page, clear the **Arrow** and **Edge** check boxes.

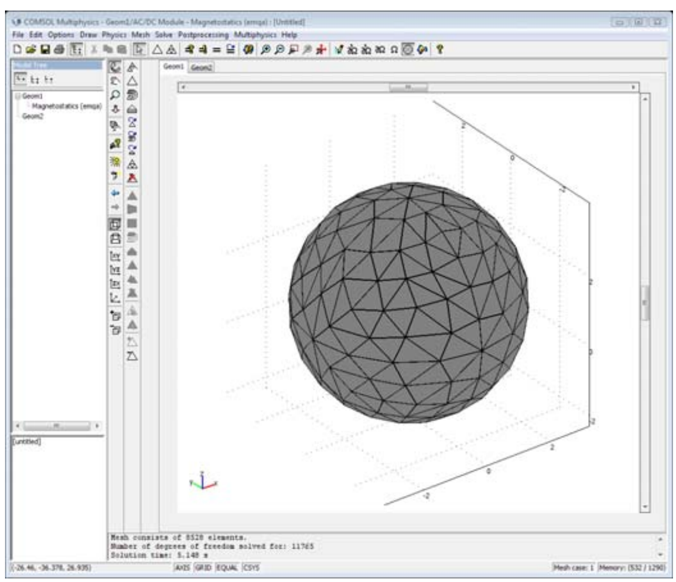
3 Select the **Plot type** called **Slice**.

4 Click **OK**.

5 Click **Zoom Extents**.



The GMG preconditioner automatically saves all mesh refinement cases and makes them accessible from the **Mesh** menu. To see the mesh, go to the **Mesh** menu and change to **Mesh case 1**. Click the **Mesh Mode** button on the Main toolbar to visualize the mesh (or select **Mesh Mode** from the **Mesh** menu):



This mesh corresponds to about 12,000 degrees of freedom.

To get an even more accurate solution now increase the order of the elements from linear to quadratic. This gives you a solution that would be computationally demanding if you had used a direct solver instead of the GMG+GMRES combination. In doing this you will create mesh case 2 in addition to the already created mesh cases 0 and 1.

1 From the **Mesh** menu select **Mesh Cases**.
2 In the **Mesh Case Settings** dialog box click **New**.
3 From the **Mesh cases** list, select mesh case **2**.
4 Select and set the check box for **Use same mesh as mesh case** to **1**.
5 Click **OK** to close the dialog box.
6 From the **Mesh** menu verify that **Mesh case 2** is the selected mesh case.
7 From the **Physics** menu select **Subdomain Settings**.
8 In the **Subdomain selection** list select Subdomain 1.
9 Click the **Element** page.
10 Change the **Predefined elements** from **Vector-Linear** to **Vector-Quadratic**.
11 Click **OK** to close the dialog box.
12 Click the **Solve** button. The GMG preconditioner now utilizes all three meshes.

The resulting mesh corresponds to about 60,000 degrees of freedom and the computations require about 300 MB of RAM.

You may get warnings about inverted mesh elements in the solver **Progress–Log** window. This means that some elements are numerically turned inside out due to the second order geometrical representation of the element edges. It typically happens at curved faces or edges in the geometry. In this model, it may happen at the rounded “corners” of the reader antenna. The problem can be avoided by using a linear geometry representation of the element edges. Go to the **Physics** menu, select **Model Settings**, and change **Geometry shape order** from **Automatic** to **Linear**. However, unless the inverted elements are located in a critical region, for example on edges where the mutual inductance is computed, or if the solver does not converge, you can safely ignore the warnings.

You can optimize the mesh even more. The mesh should accurately describe the shape of the antenna. This is not a problem for the transponder antenna because it consists of straight edges. However, the reader antenna has some curved edges, and you need to manually refine the mesh adjacent to the reader antenna by setting a maximum mesh density for the edges that makes up the reader antenna.

1 From the **Mesh** menu select **Mesh Cases**.
2 In the **Mesh Case Settings** dialog box click **New**.
3 Click **OK** to close the dialog box.
4 From the **Mesh** menu verify that **Mesh case 3** is the selected mesh case.
5 From the **Mesh** menu select **Free Mesh Parameters**.
6 Click the **Edge** tab. Keep this dialog open but move it aside for later use.

You now need to enable the visualization of all edges:

1 From the **Options** menu select **Suppress>Suppress Edges**.
2 Click the **Suppress None** button and click **OK** to close the dialog box.
3 Deselect the **Orbit/Pan/Zoom** button and enclose both the reader and transponder antenna with a rubber-band box resulting in them being highlighted in red.
4 Go back to the **Edge** page of the **Free Mesh Parameters** dialog box.
5 For the **Maximum element size for edges**, set `0.05`.
6 Click the **Global** tab.
7 Set the **Predefined mesh sizes** to **Coarse**.
8 Click the **Remesh** button.
9 Click **OK** to close the dialog box.

Later, you can visualize the interior of the mesh to see the refined mesh around the reader antenna. First you need to raise the order of the elements also for this last mesh case.

1 From the **Physics** menu, select **Subdomain Settings**.
2 In the **Subdomain selection** list select Subdomain 1.
3 Click the **Element** tab.
4 Change the **Predefined elements** from **Vector - Linear** to **Vector - Quadratic**.
5 Click **OK** to close the dialog box. The resulting mesh has about 30,000 elements and corresponds to more than 180,000 degrees of freedom. The model, at this stage, requires about 400 MB to solve.
6 From the **Solve** menu, select **Solver Parameters**.
7 Click the **Settings** button in the **Linear system solver** area.

8 In the **Linear Systems Solver Settings** dialog box, select **Preconditioner** from the tree view.

9 On the **Automatic** page, choose **Lover element order first (all)** from the **Hierarchy generation method** list. Clear the **Keep generated mesh cases** check box.

10 Click **OK** twice to close the dialog boxes.

11 Click the **Solve** button on the Main toolbar. Inverted mesh element warnings may appear.

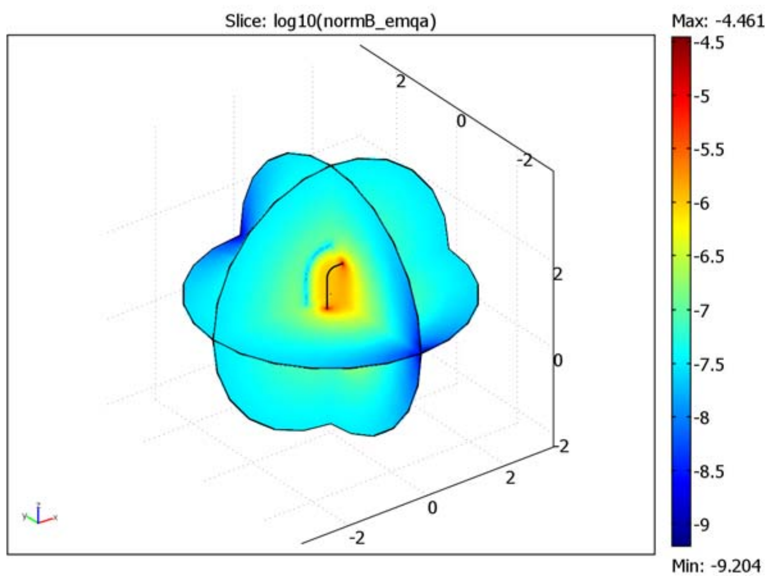
12 From the **Postprocessing** menu, select **Plot Parameters**.

13 On the **General** page, select **Slice** and **Geometry edges**, clear all other **Plot** type check boxes.

14 On the **Slice** page, for the **Slice data Expression** type `log10(normB_emqa)`. This visualizes the base-10 logarithm of the magnitude of the magnetic flux density.

15 Make sure the **Slice positioning** is set to `1`, `1`, and `1`, for the **Number of levels** in the **x**, **y**, and **z directions**.

16 Click **OK** to close the dialog box.



To get the value of the mutual inductance variable `L12`, this time for the refined mesh:

1 From the **Postprocessing** menu, select **Data Display>Global**.

2 For the **Expression to evaluate**, type `L12`.

3 Click **OK** to evaluate.

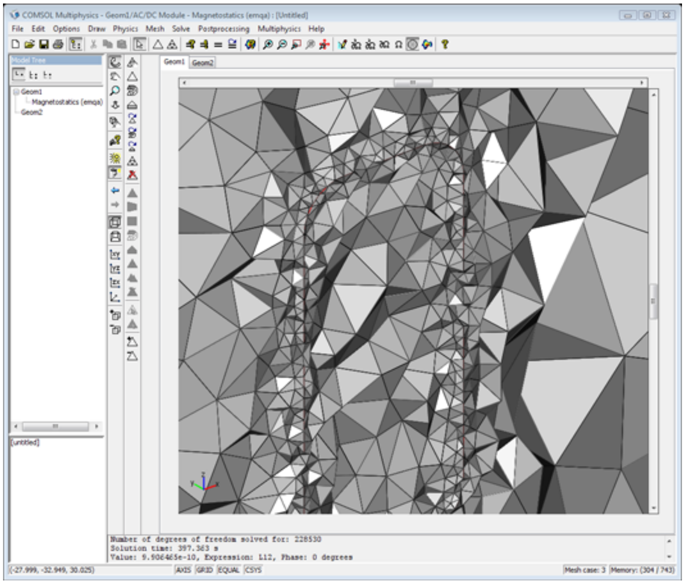
A value of **9.900337E-10** (this may vary slightly on different computers) is displayed in the history log at the bottom of the GUI window. This corresponds to a mutual inductance of about 1 nH (rounded).

ADVANCED MESH VISUALIZATION

Start by visualizing the interior of the mesh.

1 From the **Mesh** menu, select the **Mesh visualization parameters**.
2 Clear the **Boundary elements** check box.
3 Select the **Subdomain and Edge elements** check boxes.
4 Change the **Edge elements** color to a red color.
5 Change the **Subdomain elements, Element color** to a light gray color.
6 Select the **Subdomain elements, Wireframe color** check box and set it to dark gray.
7 Select the **Element selection** check box and set the **Logical expression for inclusion** field to `(y>=0.4)`.
8 Click **OK** to apply and close the dialog box.
9 Click the mesh mode button to visualize the mesh.
10 From the **Options** menu, select **Preferences**.
11 On the **Visualization** page, change the **Line width** to `1.0`.
12 Click **OK** to close the dialog box.
13 Click the **Headlight** toolbar button on the Camera toolbar.
14 Click the **Zoom Window** button and enclose the upper part of the antenna system.

Click **Zoom Extents** to start over if you zoom in too much.



The visualization shows (in red color) that the edges of the curved sections of the reader antennas are discretized so that the curvature is resolved quite well.

MAGNETIC FLUX-LINE VISUALIZATION

1 From the **Options** menu, select **Visualization/Selection Settings**.

2 On the **Rendering/Selection** page, change the **Transparency** to `1.0` (or use the **Decrease Transparency** toolbar button repeatedly).

3 Click **OK** to apply and close the dialog box.

4 From the **Postprocessing** menu, select **Plot Parameters**.

5 On the **Slice** page set the **Number of levels** in the **x** and **y direction** to `0` and leave the **z direction** setting at `1`.

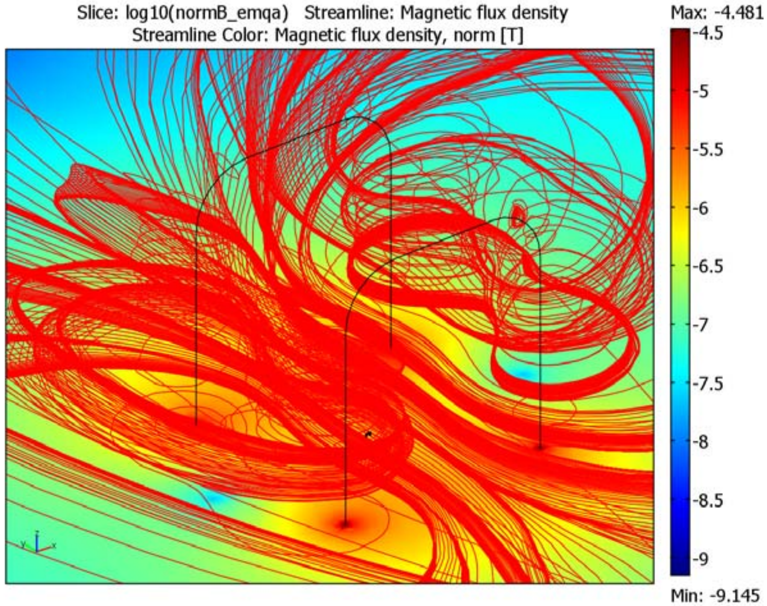
6 On the **Streamline** page, change the **Predefined quantities** to **Magnetic flux density**.

7 Change the **Number of start points** to `100`. Please note that if the graphics card on the computer has limited memory, this operation might fail. If so, try then to reduce the number from `100` to, say, `50`.

8 Select the **Streamline plot** check box in the upper-left corner.

9 Click **OK** to apply and close the dialog box.

10 Click the **Headlight** button on the Camera toolbar to switch off the light.



Time-Harmonic Simulations

The RFID tag model also makes sense to solve in time-harmonic setting where you solve for a certain frequency, or frequency-sweep, instead of the case of steady-currents and fields.

To change the model from being static to time-harmonic is fairly straightforward and is described by the following steps.

1 From the **Physics** menu, select **Properties**.
2 Change the **Analysis type** to **Time-harmonic**.
3 Change the **Potentials** (solved for) to **Electric and magnetic**.
4 Make sure **Gauge fixing** is **On**.
5 Click **OK** to close the dialog box.
6 From the **Physics** menu, select **Scalar Variables**.
7 Change the **Frequency** to the desired one, say `50000` Hz. Note that the frequency variable is available as `nu_emqa` (by default).

8 Click **OK** to close the dialog box.

The solver settings change when you change the analysis type. To change to GMRES with GMG, adjust the corresponding **Solver Parameters**. The default element type for this analysis type is linear elements: linear vector elements for the magnetic vector potential and linear Lagrange (isoparametric) elements for the electric potential and the gauge-fix variable psi. In order to change the element order for the mesh cases, select each mesh case you want to change the settings for and adjust the settings on the **Element** page of the **Subdomain Settings** dialog box. Please note that if you change the defaults, from linear to quadratic for this example, you might run out of memory unless you are using a computer with 2 GB of RAM or more.

These are all the necessary changes for solving a time-harmonic problem. Please note that because this problem is induction dominated, the computed value of the inductance, `L12`, do not change significantly unless the frequency is very high.

Increasing the frequency of the RFID system does increase radiation losses and eventually artificial reflections from the truncation sphere will appear. (So far we have ignored radiation losses.) When this becomes a problem you need to add absorbing layers or boundary conditions to the outside of the domain. In such a case, you also need to use the Electromagnetic Waves, Harmonic Propagation application mode found in the RF Module. Such high frequencies are unusual for an RFID system. However, if you apply the above modeling technique to another type of wire-antenna system you might need to take this into consideration. See the RF Module documentation for more information on absorbing layers (called perfectly matched layers or PMLs) and boundary conditions.

INDEX