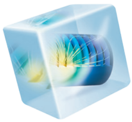


Introduction to AC/DC Module

VERSION 4.3

COMSOL

Introduction to the AC/DC Module

Version: May 2012 COMSOL 4.3

Contact Information

Visit www.comsol.com/contact for a searchable list of all COMSOL offices and local representatives. From this web page, search the contacts and find a local sales representative, go to other COMSOL websites, request information and pricing, submit technical support queries, subscribe to the monthly eNews email newsletter, and much more.

If you need to contact Technical Support, an online request form is located at www.comsol.com/support/contact.

Other useful links include:

- Technical Support www.comsol.com/support
- Software updates: www.comsol.com/support/updates
- Online community: www.comsol.com/community
- Events, conferences, and training: www.comsol.com/events
- Tutorials: www.comsol.com/products/tutorials
- Knowledge Base: www.comsol.com/support/knowledgebase

Part No. CM020104

Contents

Introduction 5
The Use of the AC/DC Module. 6
The AC/DC Module Physics Guide 10
Physics List by Space Dimension and Preset Study Type15
Opening the Model Library. 16
Tutorial Example: Modeling a 3D Inductor 17

Introduction

The AC/DC Module is used by engineers and scientists to understand, predict, and design electric and magnetic fields in statics and low-frequency applications. Simulations of this kind result in more powerful and efficient products and engineering methods. It allows its users to quickly and accurately predict electromagnetic field distributions, electromagnetic forces, and power dissipation in a proposed design. Compared to traditional prototyping, COMSOL helps to lower costs and can evaluate and predict entities that are not directly measurable in experiments. It also allows the exploration of operating conditions that would destroy a real prototype or be hazardous.

The AC/DC Module includes stationary and dynamic electric and magnetic fields in two-dimensional and three-dimensional spaces along with traditional circuit-based modeling of passive and active devices. All modeling formulations are based on Maxwell's equations or subsets and special cases of these together with material laws like Ohm's law for charge transport. The modeling capabilities are accessed via a number of predefined user interfaces, referred to as AC/DC interfaces, which allow you to set up and solve electromagnetic models. The AC/DC interfaces cover electrostatics, DC current flow, magnetostatics, AC and transient current flow, AC and transient magnetodynamics, and AC electromagnetic (full Maxwell) formulations.

Under the hood, the AC/DC interfaces formulate and solve the differential form of Maxwell's equations together with initial and boundary conditions. The equations are solved using the finite element method with numerically stable edge element discretization in combination with state-of-the-art algorithms for preconditioning and solution of the resulting sparse equation systems. The results are presented in the graphics window through predefined plots of electric and magnetic fields, currents and voltages or as expressions of the physical quantities that you can define freely, and derived tabulated quantities (for example resistance, capacitance, inductance, electromagnetic force, and torque) obtained from a simulation.

The work flow in the module is straightforward and is described by the following steps: define the geometry, select materials, select a suitable AC/DC interface, define boundary and initial conditions, define the finite element mesh, select a solver, and visualize the results. All these steps are accessed from the COMSOL Desktop. The solver step is usually carried out automatically using the default settings, which are already tuned for each specific interface.

The AC/DC Module Model Library describes the AC/DC interfaces and the available features using tutorial and benchmark examples for the different formulations. The library contains models for industrial equipment and devices, tutorial models, and benchmark models for verification and validation of the AC/DC interfaces. See "Opening the Model Library" on page 16 to start exploring now.

This guide is intended to give you a jump start to your modeling work. It contains examples of the typical use of the module, a list of all the AC/DC interfaces including a short description for each interface, and a tutorial example that introduces the modeling workflow.

The Use of the AC/DC Module

The AC/DC Module can model electric, magnetic, and electromagnetic fields in statics and low-frequency applications. The latter means that it covers the modeling of devices that are up to about 0.1 electromagnetic wavelengths in size. Thus, it can be used to model nanodevices up to optical frequencies or for human size devices operated at frequencies up to 10 MHz.

AC/DC simulations are frequently used to extract circuit parameters. Figure 1 shows the electric potential and the magnetic flux lines for a micro-scale square inductor, used for LC bandpass filters in microelectromechanical systems (MEMS). A DC voltage is applied across the electrically conducting square shaped spiral, resulting in an electric current flow that in turn generates a magnetic field through and around the device.

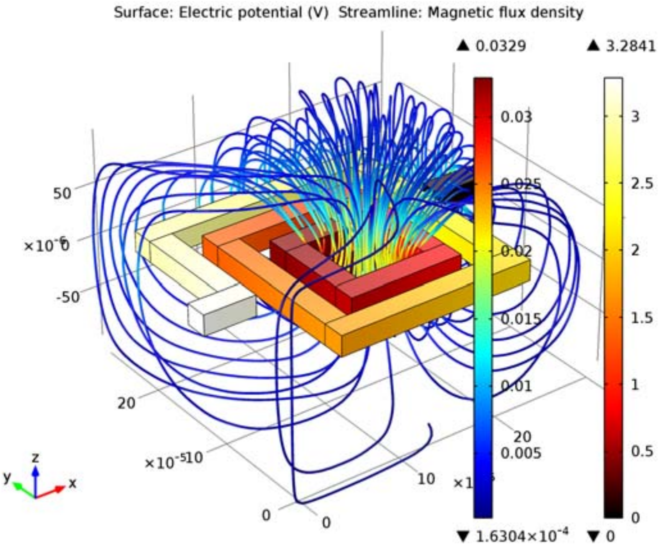


Figure 1: Electric potential distribution and magnetic flux lines for a MEMS inductor.

The distribution and strength of electric and magnetic fields arising from applied current and voltage can be translated into resistance, capacitance, and inductance values for an equivalent circuit model that is easy to understand and analyze from both qualitative and quantitative perspectives. This kind of understanding in terms of a simplified model is usually the first step when creating or improving a design.

Another common use of this module is to predict forces and motion in electric motors and actuators of a wide range of scales.

In the next three figures (Figure 2, Figure 3, and Figure 4), the dynamics of a magnetic brake is shown. The brake consists of a disc of conductive material and a permanent magnet. The magnet generates a constant magnetic field, in which the disc is rotating.

When a conductor moves in a magnetic field it induces currents, and the Lorentz forces from the currents counteracts the spinning of the disc.

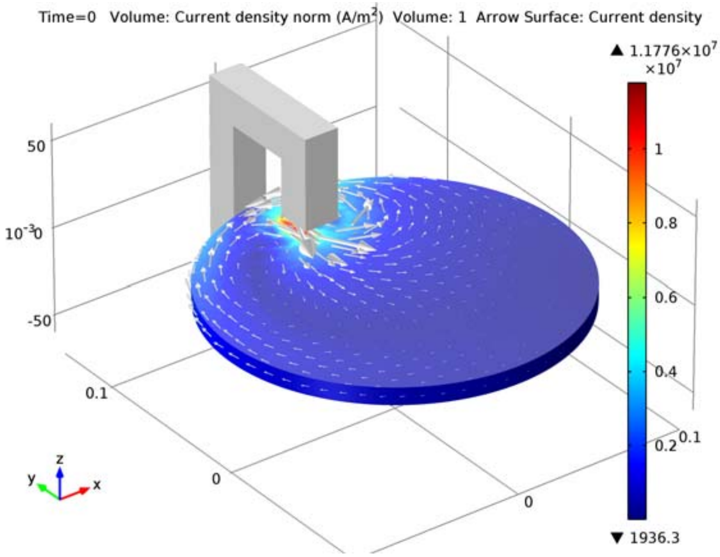


Figure 2: The eddy current magnitude and direction in the spinning disc of a magnetic brake.

The braking torque is computed from the magnetic flux and eddy current distributions. It is then fed into a rigid body model for the dynamics of the spinning disc that is solved simultaneously with the electromagnetic model.

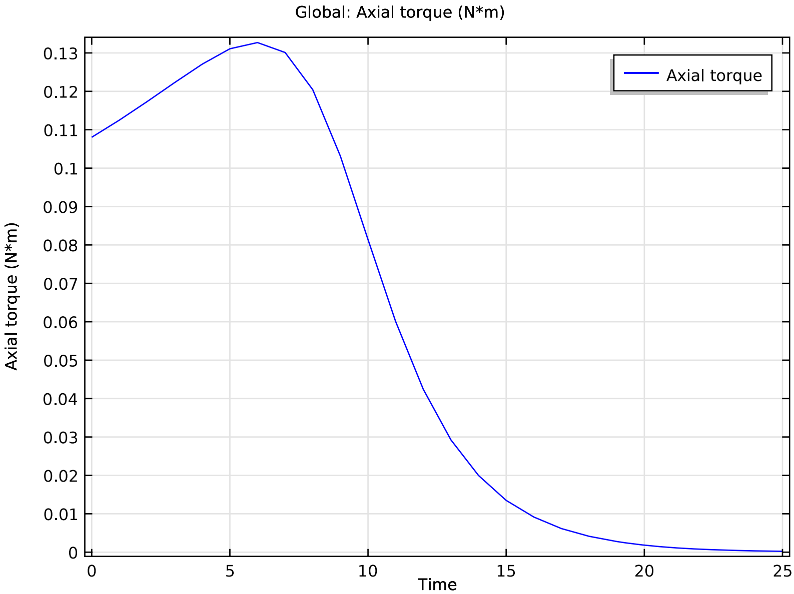


Figure 3: The time evolution of the braking torque.

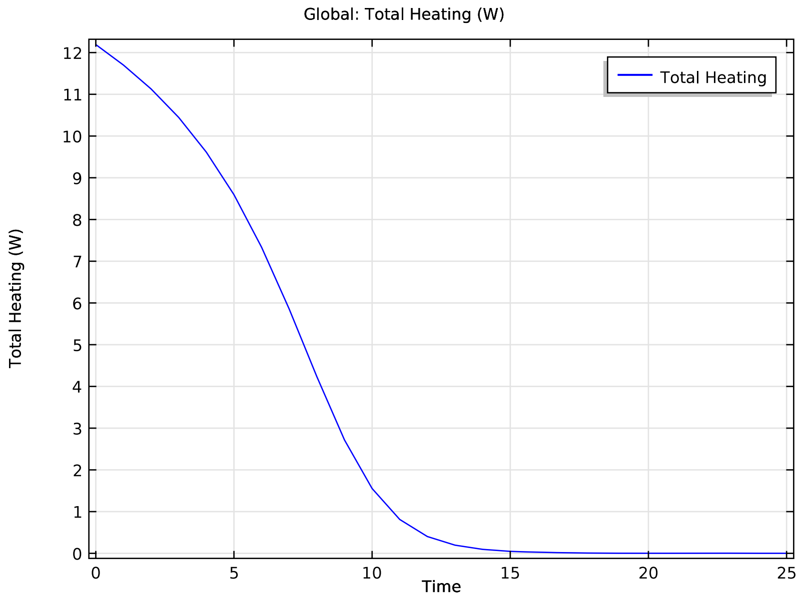


Figure 4: The time evolution of the angular velocity.

The AC/DC Module includes a vast range of tools to evaluate and export results, for example, the evaluation of force, torque, and lumped circuit parameters like resistance, capacitance, inductance, impedance, and scattering matrices (S-parameters). Lumped parameters can also be exported in the Touchstone file format.

The combination of fully distributed electromagnetic field modeling and simplified circuit based modeling is an ideal basis for design, exploration, and optimization. More complex system models can be exploited using circuit-based modeling while maintaining links to full field models for key devices in the circuit allowing for design innovation and optimization on both levels.

The AC/DC Module Interfaces

The AC/DC physics interfaces are based upon Maxwell's equations or subsets and special cases of these together with material laws. Different subsets of Maxwell's equations in combination with special material laws result in different electric, magnetic, or electromagnetic formulations. In the module, these laws of physics are translated by the AC/DC interfaces to sets of partial differential equations with corresponding initial and boundary conditions.

An AC/DC interface defines a number of features. Each feature represents an operation that describes a term or condition in the underlying Maxwell-based formulation. Such a term or condition may be defined in a geometric entity of the model, such as a domain, boundary, edge (for 3D models), or point.

Figure 5 on the next page uses the Capacitor Tunable model (found in the AC/DC Module Model Library) to show the Model Builder window and the settings window for the selected Charge Conservation 1 feature node. The Charge Conservation 1 node also adds the terms to the model equations to a selected geometrical domain representing the electrostatics domain in the model.

Furthermore, and shown in Figure 5, the Charge Conservation 1 node may link to the Materials node to obtain physical properties such as relative permittivity, in this case the relative permittivity of a user-defined dielectric. The properties, defined by the Dielectric material node, may be functions of the modeled physical quantities, such as temperature. In the same fashion, the Zero Charge 1 boundary condition feature adds the natural boundary conditions that limit the Electrostatics domain.

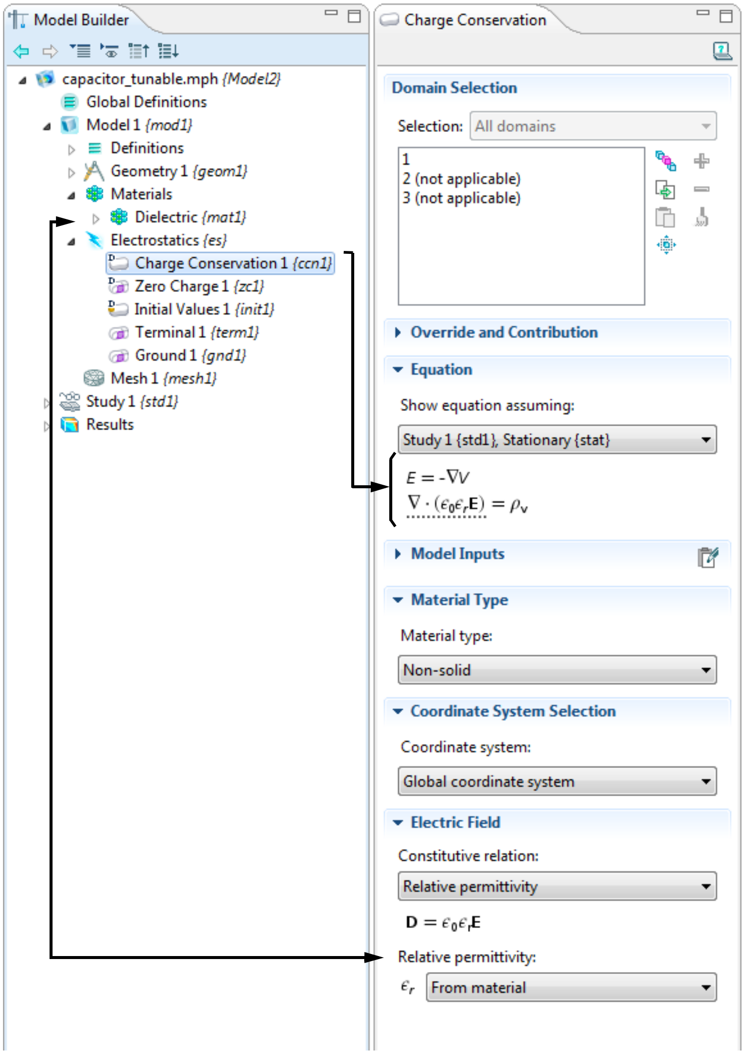


Figure 5: The Model Builder window (to the left), and the Charge Conservation 1 settings window for the selected feature node (to the right). The Equation section in the settings window shows the model equations and the terms added to the model equations by the Charge Conservation 1 node. The added terms are underlined with a dotted line. The text also explains the link between the Dielectric node and the values for the Relative permittivity.

The AC/DC Module includes a number of AC/DC interfaces () for different types of electric and/or magnetic modeling. Figure 6 includes the AC/DC interfaces as well as those under the Heat Transfer folder, which contains the Induction Heating interface. Also see "Physics List by Space Dimension and Preset Study Type" on page 15. A brief overview of the AC/DC interfaces follows.

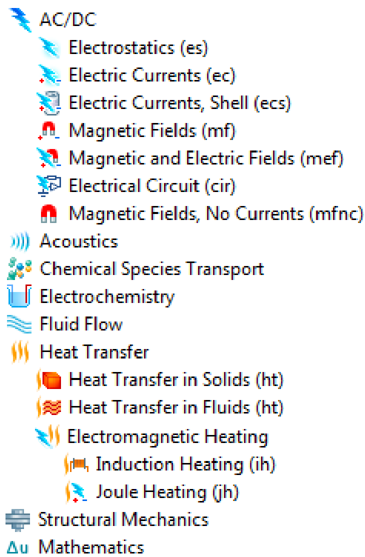


Figure 6: The AC/DC Module physics as displayed in the Model Wizard. Note that this is for 3D models. The Rotating Machinery, Magnetic interface is also available in 2D.

ELECTROSTATICS

The Electrostatics interface ( ) solves a charge conservation equation for the electric potential given the spatial distribution of the electric charge. It is used primarily to model charge conservation in dielectrics under static conditions. The interface applies also under transient conditions but then it is usually combined with a separate transport model for single species or multispecies charge transport. Such transport models can be found in the Chemical Reaction Engineering Module and in the Plasma Module. Some typical applications that are simulated in the interface are capacitors, dielectric sensors, and bushings for high voltage DC insulation.

ELECTRIC CURRENTS

The Electric Currents interface ( ) is used to model DC, AC, and transient electric current flow in conductive and capacitive media. The interface solves a current conservation equation for the electric potential. Some examples of its use are designing busbars for DC current distribution, computing current distributions in impressed current cathodic corrosion protection, and designing AC capacitors.

Electric Currents, shell

The Electric Currents, Shell interface ( ) is available in 2D, 2D axisymmetric, and 3D geometries. It is used to model DC, AC, and transient electric current flow confined to conductive and capacitive thin current-conducting shells of fixed or varying thickness. The interface solves a boundary current conservation equation for the electric potential. Modeling ground return current flow in the hull of a ship or in the body of a car are two examples of simulations that can be performed using this interface.

Magnetic Fields

The Magnetic Fields interface ( ) solves Ampère’s law for the magnetic vector potential. It is used to model magnetostatics, AC, and transient magnetodynamics. Magnets, magnetic actuators, electric motors, transformers, induction based nondestructive testing, and eddy current generation are typical applications for this interface. It supports both linear media and media with magnetic saturation.

Magnetic and Electric Fields

The Magnetic and Electric Fields interface ( ) is used to model magnetostatics and AC magnetodynamics. It solves Ampère’s law for the magnetic vector potential together with a current conservation equation for the electric potential. The application areas are mostly the same as for the Magnetic Fields interface. The addition of the electric potential makes it more flexible as also electric boundary conditions are available.

Magnetic Fields, No Currents

The Magnetic Fields, No Currents interface ( ) is used to efficiently model magnetostatics in current free regions, for example, when designing permanent magnet-based devices. It solves a magnetic flux conservation equation for the magnetic scalar potential. The interface supports both linear media and media with magnetic saturation.

Electrical Circuit

The Electrical Circuit interface ( ) has the equations to model electrical circuits with or without connections to a distributed fields model, solving for the voltages, currents, and charges associated with the circuit elements. Circuit models can contain passive elements like resistors, capacitors, and inductors as well as active elements such as diodes and transistors.

Rotating Machinery, Magnetic

The Rotating Machinery, Magnetic interface ( ) combines the magnetic fields (magnetic vector potential) and magnetic fields, no currents (scalar magnetic potential) formulations with a selection of predefined frames for prescribed rotation or rotation velocity—it shares most of its features with the Magnetic Fields interface. This interface requires that the geometry is created as an assembly from individual parts for the rotor and stator.

Induction Heating

The Induction Heating multiphysics interface ( ) combines all features from the Magnetic Fields interface in the time-harmonic formulation with the Heat Transfer interface to model induction and eddy current heating. The predefined interaction adds the electromagnetic losses from the magnetic field as a heat source. This interface is based on the assumption that the magnetic cycle time is short compared to the thermal time scale (adiabatic assumption).

Physics List by Space Dimension and Preset Study Type

The table below list the interfaces available specifically with this module in addition to the COMSOL Multiphysics basic license.

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
AC/DC				
Electrostatics*		es	all dimensions	stationary; time dependent
Electric Currents*		ec	all dimensions	stationary; frequency domain; time dependent; small signal analysis, frequency domain
Electric Currents - Shell		ecs	3D, 2D, 2D axisymmetric	stationary; frequency domain; time dependent; small signal analysis, frequency domain
Magnetic Fields*		mf	3D, 2D, 2D axisymmetric	stationary; frequency domain; time dependent; small signal analysis, frequency domain; coil current calculation

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Magnetic and Electric Fields		mef	3D, 2D, 2D axisymmetric	stationary; frequency domain; small signal analysis, frequency domain
Magnetic Fields, No Currents		mfnc	all dimensions	stationary; time dependent
Rotating Machinery, Magnetic		rmm	3D, 2D	stationary; time dependent
Electrical Circuit		cir	Not space dependent	stationary; frequency domain; time dependent
Electromagnetic Heating				
Induction Heating		ih	3D, 2D, 2D axisymmetric	stationary; frequency domain; time dependent; frequency stationary; frequency transient

* This is an enhanced interface, which is included with the base COMSOL package but has added functionality for this module.

Opening the Model Library

To open an AC/DC Module Model Library model, select **View > Model Library** from the main menu in COMSOL Multiphysics. In the Model Library window that opens, expand the AC/DC Module folder and browse or search the contents. Click **Open Model and PDF** to open the model in COMSOL Multiphysics and a PDF to read background theory about the model including the step-by-step instructions to build it. The MPH-files in the COMSOL model libraries can have two formats—Full MPH-files or Compact MPH-files.

- Full MPH-files, including all meshes and solutions. In the Model Library these models appear with the icon. If the MPH-file's size exceeds 25MB, a tip with the text "Large file" and the file size appears when you position the cursor at the model's node in the Model Library tree.
- Compact MPH-files with all settings for the model but without built meshes and solution data to save space on the DVD (a few MPH-files have no solutions for other reasons). You can open these models to study the settings and to mesh and re-solve the models. It is also possible to download the full versions—with meshes and solutions—of most of these models through Model Library Update. In the Model Library these models appear with the icon.

The next section uses a model from the Model Library. Go to "Tutorial Example: Modeling a 3D Inductor" to get started.

Tutorial Example: Modeling a 3D Inductor

Inductors are used in many applications for low-pass filtering or for impedance matching of predominantly capacitive loads. They are used in a wide frequency range from near static up to several MHz. An inductor usually has a magnetic core to increase the inductance, while keeping its size small. The magnetic core also reduces the electromagnetic interference with other devices as the magnetic flux tends to stay within it. Because there are only crude analytical or empirical formulas available to calculate impedances, computer simulations or measurements are necessary during the design stage. In general, inductor modeling is more complex than modeling resistors and capacitors but similar principles apply. Using an external CAD software program to design and draw the model, the geometry is imported into the AC/DC Module for static and frequency domain analysis. The inductor geometry is shown in Figure 7.

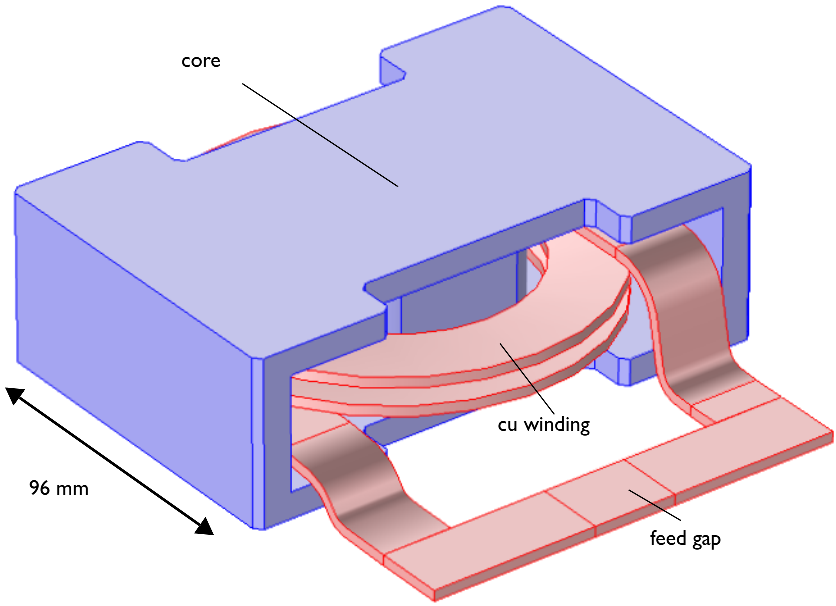


Figure 7: The inductor geometry.

First a magnetostatic simulation is performed to get the DC inductance. At low frequencies capacitive effects are negligible. A relevant equivalent circuit model is an ideal inductor in a series with an ideal resistor. The inductance and the resistance are both computed in the magnetostatic simulation. At a high frequency, capacitive effects become significant. The equivalent circuit model involves connecting an ideal capacitor in parallel with the DC circuit. The circuit parameters are obtained by analyzing the frequency dependent impedance obtained from a frequency domain simulation. In this tutorial, the AC analysis is done up to the point when the frequency dependent impedance is computed.

These step-by-step instructions guide you through the detailed modeling of an inductor in 3D. The module also has a physics interface to model electrical circuits, which is detailed in the magnetostatic part of this model. The first step is to perform a DC simulation.

MODEL WIZARD

1. Double-click the COMSOL Multiphysics icon on the desktop.
2. In the **Model Wizard**, click the **3D** button and click **Next**.
3. In the **Add Physics** tree under **AC/DC**, select **Magnetic Fields (mf)**.
4. Click **Add Selected** then click **Next**.
5. In the **Studies** tree, select **Preset Studies>Stationary**.
6. Click **Finish**.

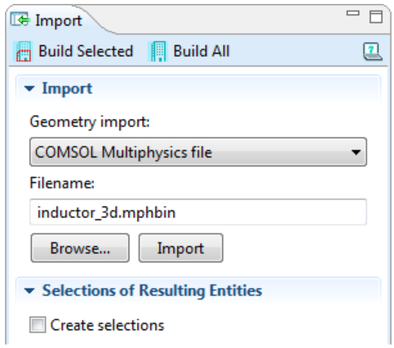
GEOMETRY 1

The main geometry is imported from a file. Air domains are typically not part of a CAD geometry, so these are added to the model. For convenience, three additional domains are defined in the CAD file and then used to define a narrow feed gap where an excitation is applied.

Import 1

1. Under **Model 1**, right-click **Geometry 1** and select **Import**.
2. In the settings window under **Import**, click **Browse**. Then navigate to your COMSOL Multiphysics installation folder, locate the subfolder **ACDC_Module\Inductive_Devices_and_Coils**, select the file `inductor_3d.mphbin` and click **Open**.

Note: The location of the files used in this exercise varies based on the installation. For example, if the installation is on your hard drive, the file path might be similar to `C:\Program Files\COMSOL43\models\`.

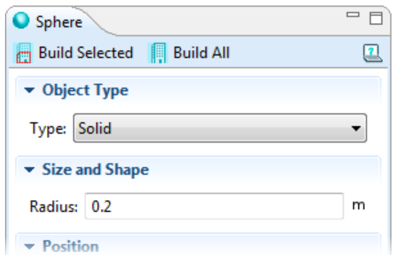


3 Click **Import**.

Sphere 1

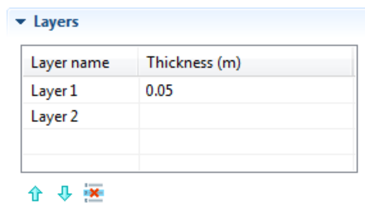
1 In the **Model Builder**, right-click **Geometry 1** and choose **Sphere**.

2 Go to the **Sphere** settings window. Under **Size and Shape** in the **Radius** field, enter `0.2`.



3 Click to expand the **Layers** section. In the associated table, under **Thickness** enter `0.05`.

4 Click the **Build All** button.

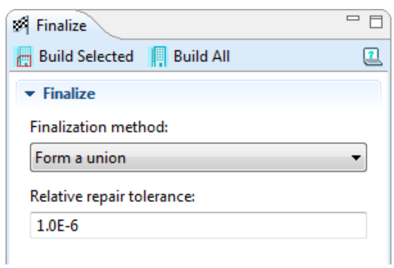


Form Union

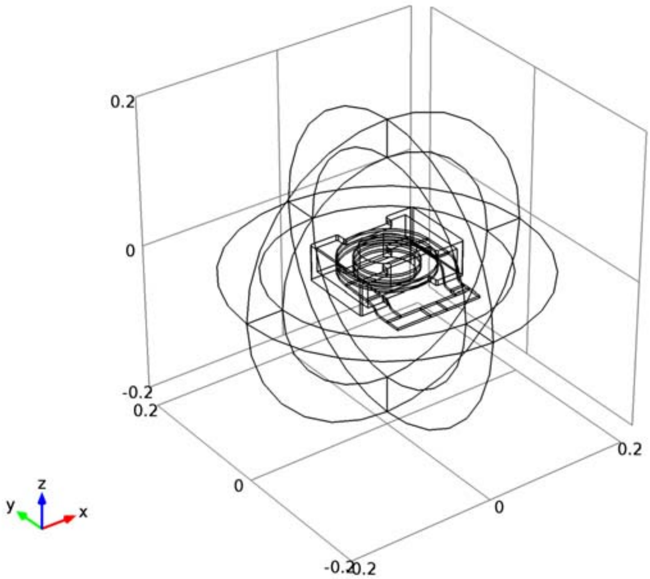
1 In the **Model Builder** under **Geometry 1**, click **Form Union**.

2 In the **Finalize** settings window, click **Build All**.

3 On the **Graphics** window toolbar, click the **Zoom Extents** button and then the **Wireframe Rendering** button.



The geometry should match this figure.

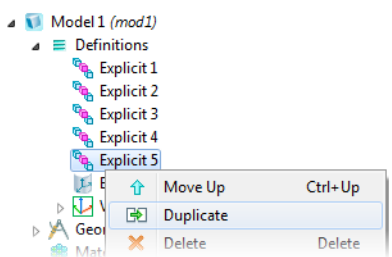


DEFINITIONS - SELECTIONS

Use the Selection feature available under the Definitions node to make the set up of materials and physics easier. Start by defining the domain group for the inductor winding and continue by adding other useful selections. These steps illustrate how to set up the Explicit nodes and rename the geometric selections accordingly.

1 In the **Model Builder** under **Model 1**, right-click **Definitions** and choose **Selections>Explicit**.

2 Repeat Step 1 and add a total of six (6) **Explicit** nodes. Or right-click the first **Explicit** node and select **Duplicate**.



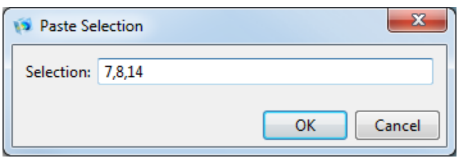
The following steps are done one at a time for each node using the table below.

3 In the **Model Builder** click an **Explicit** node to open its settings window.

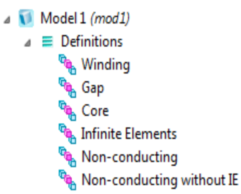
4 In the settings window under **Geometric Scope**, select **Domain** from the **Geometric entity level** list. Use the table as a guide, and in the **Graphics** window, select the domains as indicated. Then press F2 and rename the node.

DEFAULT NODE NAME	SELECT THESE DOMAINS	NEW NAME FOR THE NODE
Explicit 1	7, 8, and 14	Winding
Explicit 2	9	Gap
Explicit 3	6	Core
Explicit 4	1–4 and 10–13	Infinite Elements
Explicit 5	1–6 and 9–13	Non-conducting
Explicit 6	5,6, and 9	Non-conducting without IE

Note: There are many ways to select geometric entities. When you know the domain to add, such as in this exercise, you can click the **Paste Selection** button and enter the information in the **Selection** field. In this example for the **Explicit 1** node, enter `7,8,14` in the **Paste Selection** window. For more information about selecting geometric entities in the **Graphics** window, see the *COMSOL Multiphysics User's Guide*.



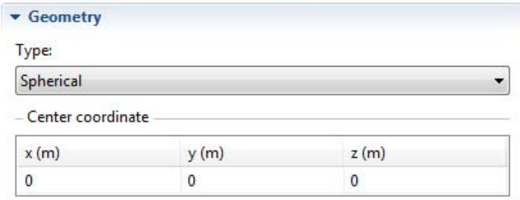
5 After renaming all the nodes under Definitions, the sequence of nodes should match this figure.



DEFINITIONS - INFINITE ELEMENTS

Use the Infinite Element feature available under the Definitions node to emulate an infinite open space surrounding the inductor. The infinite element coordinate scaling is recognized by any physics interface supporting it - other physics interfaces are automatically disabled in the infinite element domains.

1 In the **Model Builder** under **Model 1**, right-click **Definitions** and choose **Infinite Element Domain**.
2 Go to the **Infinite Elements** settings window. Under **Domain Selection** select **Infinite Elements** from the **Selection** list.
3 Under **Geometry** from the **Type** list, select **Spherical**.

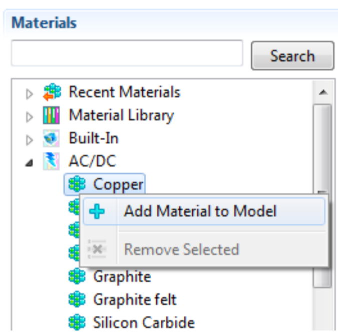


MATERIALS

Now define the material settings for the different parts of the inductor. Use the Selections defined in the previous section.

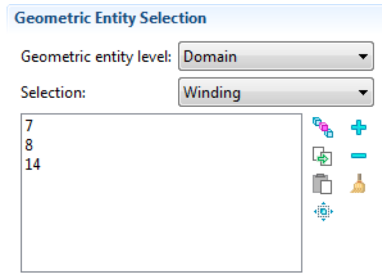
Copper

1 From the main menu, select **View>Material Browser**.
2 Go to the **Material Browser**. In the **Materials** tree under **AC/DC**, right-click **Copper** and choose **Add Material to Model** from the menu.



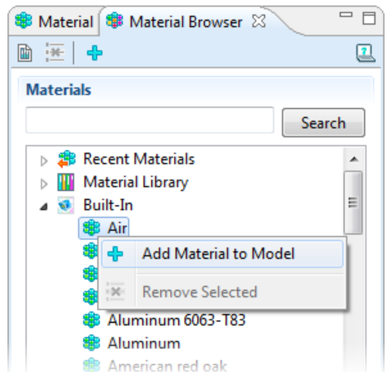
3 In the **Model Builder**, click **Copper**.

4 Go to the **Material** settings window. Under **Geometric Entity Selection** select **Winding** from the **Selection** list.



Air

1 Go to the **Material Browser**. In the **Materials** tree under **Built-In** right-click **Air** and choose **Add Material to Model** from the menu.



2 In the **Model Builder**, click **Air**.

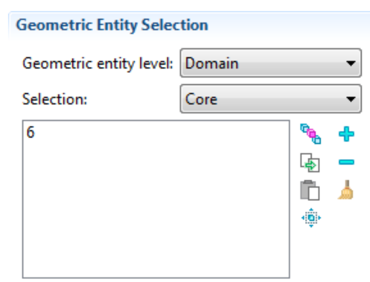
3 Go to the **Material** settings window. Under **Geometric Entity Selection** select **Non-conducting** from the **Selection** list.

USER-DEFINED MATERIAL 3

The core material is not included in the material library so it is entered as a user-defined material.

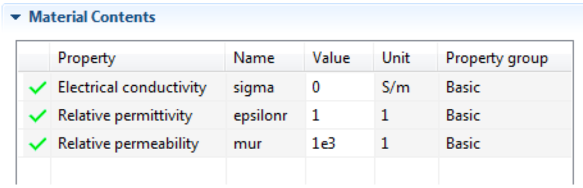
1 In the **Model Builder**, right-click **Materials** and choose **Material** .

2 Go to the **Material** settings window. Under **Geometric Entity Selection** select **Core** from the **Selection** list.



3 Under **Material Contents** in the table, enter these settings in the **Value** column:

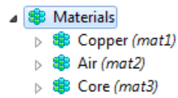
- 0 for the **Electrical conductivity**
- 1 for the **Relative permittivity**
- 1e3 for the **Relative permeability**


Material Contents

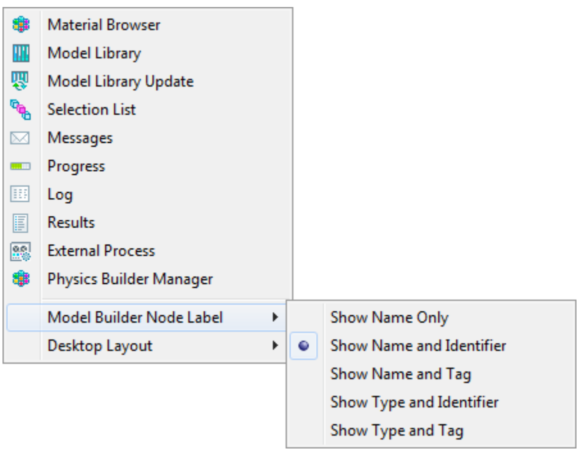
Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	0	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic
Relative permeability	mur	1e3	1	Basic

4 Press F2 and choose **Rename**. Go to the **Rename Material** dialog box and enter `Core` in the **New name** field.

5 Click **OK**. The node sequence under Materials should match the figure to the right.

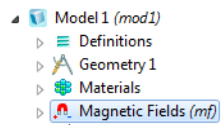


To view the **Identifier** (mat1, mat2, mat3) next to the material node name, select **View>Model Builder Node Label** and select one of the options. This changes what combinations of labels are displayed in the Model Builder next to each node.



MAGNETIC FIELDS (MF)

The model is solved everywhere except in the feed gap region. By keeping this void, its default boundary conditions support surface currents closing the current loop. In inductor modeling it is crucial to respect current conservation as this is a law of nature (Ampère's law).



1 In the **Model Builder** under **Model 1**, click **Magnetic Fields**.
2 In the **Magnetic Fields** settings window, select domains 1–8 and 10–14 only (all domains except 9). Or click the **Paste Selection** button and enter 1–8, 10–14 in the **Selection** field.

Single-Turn Coil Domain 1 and Boundary Feed 1

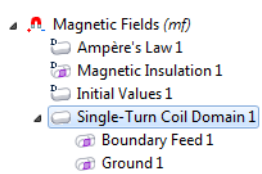
A dedicated feature is added for the solid copper winding of the inductor. This feature makes it easier to excite the model in various ways.

1 Right-click **Magnetic Fields** and choose the domain setting **Single-Turn Coil Domain**.
2 In the **Single-Turn Coil Domain** settings window, under **Domain Selection**, choose **Winding** from the **Selection** list.
3 Right-click **Single-Turn Coil Domain 1** and at the boundary level choose **Boundary Feed**.
4 In the **Boundary Feed** settings window, to the right of the **Boundary Selection** list, click the **Clear Selection** button.
5 Click the **Paste Selection** button and enter 58 in the **Selection** field. Click **OK**.

Ground 1

1 In the **Model Builder**, right-click **Single-Turn Coil Domain 1** and choose the boundary condition **Ground**.
2 In the **Ground** settings window, to the right of the **Boundary Selection** list, click the **Clear Selection** button.
3 Click the **Paste Selection** button and enter 79 in the **Selection** field. Click **OK**.

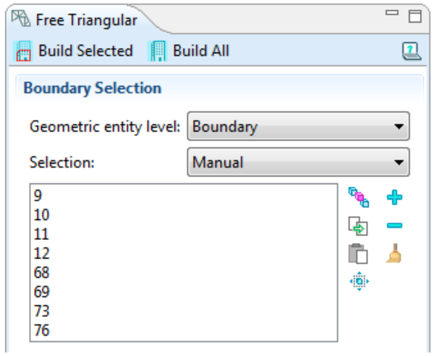
The node sequence under Magnetic Fields should at this point match the figure to the right.



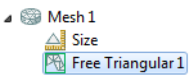
MESH 1

The steep radial scaling of the infinite elements region requires a swept mesh to maintain a reasonably effective element quality.

Free Triangular 1 and Size



1. In the **Model Builder** under **Model 1**, right-click **Mesh 1** and choose **More Operations>Free Triangular**.
2. Select Boundaries 9–12, 68, 69, 73, and 76 only. Or click the **Paste Selection** button and enter 9–12, 68, 69, 73, 76 in the **Paste Selection** field.
3. In the **Model Builder** under **Mesh 1**, click **Size**. In the **Size** settings window under **Element Size**, click the **Predefined** button. From the list select **Coarser**.
4. Click to expand the **Element Size Parameters** section. In the **Minimum element size** field, enter `0.005`.
5. In the **Model Builder**, right-click **Free Triangular 1** and select **Build Selected**.



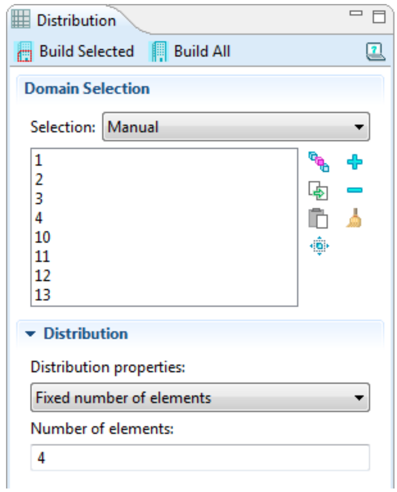
Swept 1 and Distribution 1

1. Right-click **Mesh 1** and choose **Swept**.
2. Go to the **Swept** settings window. Under **Domain Selection**:
   - From the **Geometric entity level** list, select **Domain**.
   - From the **Selection** list, select **Infinite Elements**.

3 Right-click **Swept 1** and choose **Distribution**.

4 Go to the **Distribution** settings window. Under **Distribution** in the **Number of elements** field, enter 4.

5 Click **Build Selected**.

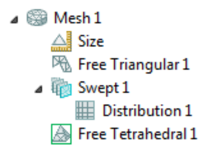


Free Tetrahedral 1

1 Right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Model Builder**, right-click **Free Tetrahedral 1** and choose **Build Selected**.

The final node sequence under **Mesh** should match the figure.



STUDY 1

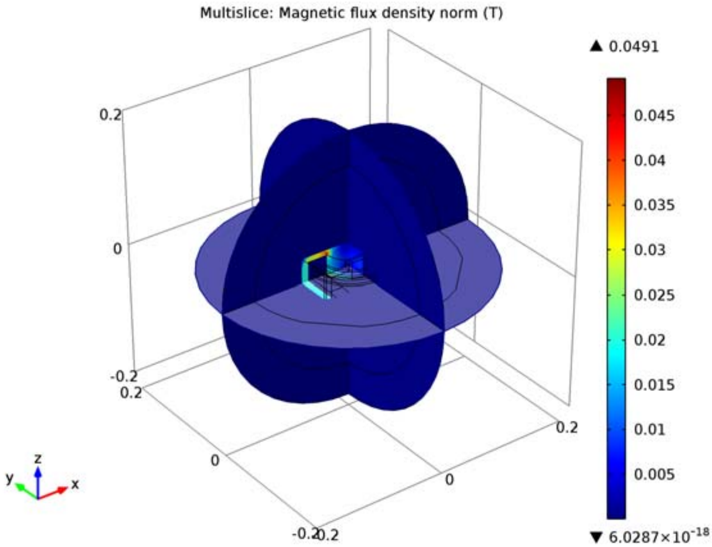
The magnetostatic model is now ready to solve.

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

RESULTS

Magnetic Flux Density

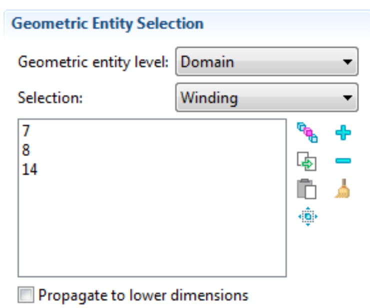
The default plot group shows the magnetic flux density norm. At this point it is important to consider whether the results make sense and to try and detect any modeling errors.



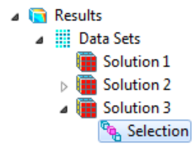
DATA SETS AND SELECTIONS

By adding Selections and adjusting the Data Sets under the **Results** node you can create more interesting plots.

1. In the **Model Builder**, expand the **Results>Data Sets** node. Right-click **Solution 1** and choose **Duplicate**.
2. Right-click **Solution 2** and choose **Add Selection**.
3. Go to the **Selection** settings window. Under **Geometric entity selection**:
   - From the **Geometric entity level** list, select **Domain**.
   - From the **Selection** list, select **Winding**.
4. Right-click **Solution 1** and choose **Duplicate**.



5 Right-click **Solution 3** and choose **Add Selection**. A **Selection** node is added to the Model Builder.



6 Go to the **Selection** settings window. Under **Geometric Entity Selection**:

- From the **Geometric entity level** list, select **Domain**.
- From the **Selection** list, select **Core**.

3D Plot Group 2

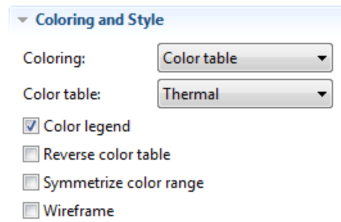
1 In the **Model Builder**, right-click **Results** and choose **3D Plot Group**.

2 Right-click **3D Plot Group 2** and choose **Volume**.

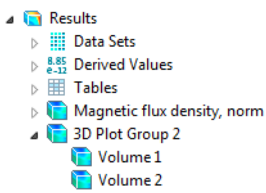
3 Go to the **Volume** settings window. Under **Data** from the **Data set** list, select **Solution 2**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Magnetic Fields>Coil parameters>Coil voltage (mf.VCoil)**.

5 Under **Coloring and Style** from the **Color table** list, select **Thermal**.

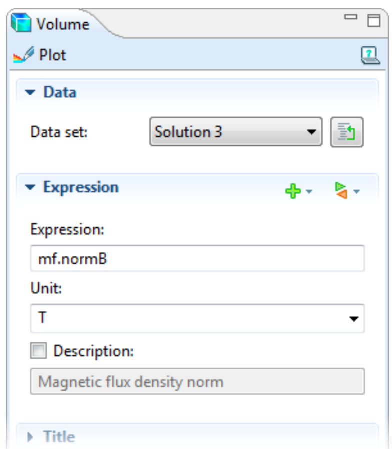


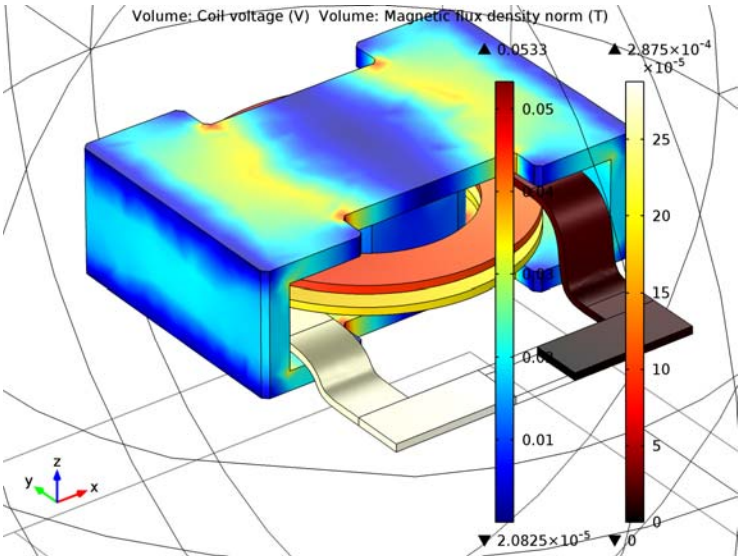
6 In the **Model Builder**, right-click **3D Plot Group 2** and choose **Volume**. A **Volume 2** node is added to the **Model Builder**.



7 Go to the **Volume** settings window. Under **Data** from the **Data set** list, select **Solution 3**.

8 Click the **Plot** button. On the **Graphics** window toolbar, click the **Zoom In** button twice.

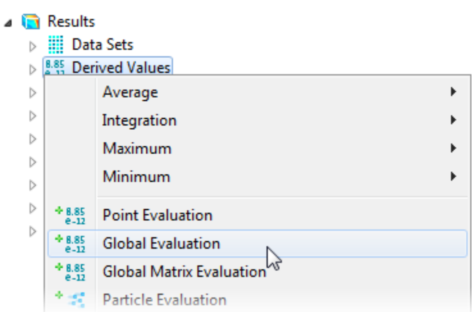




DERIVED VALUES

The coil inductance and resistance are available as predefined variables, which are displayed as follows.

1 In the **Model Builder** under **Results**, right-click **Derived Values** and choose **Global Evaluation**.



2 In the **Global Evaluation** settings window, locate the **Expression** section.

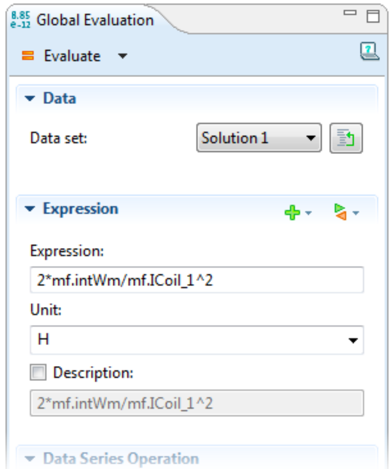
3 In the **Expression** edit field, enter `mf.VCoil_1/mf.ICoil_1`.

4 Click the **Evaluate** button. The results are available in the **Results** window table located in the lower right side of the COMSOL Desktop. Or select **View>Results** from the main menu to open the **Results** window.

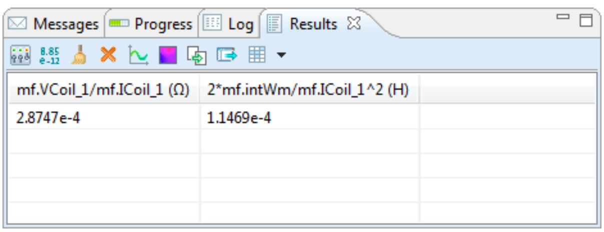
5 In the **Model Builder**, right-click **Derived Values** and choose **Global Evaluation**.

6 In the **Global Evaluation** settings window, in the **Expression** field, enter `2*mf.intWm/mf.ICoil_1^2`.

7 Click the **Evaluate** button and compare the results in the **Results** window.



You can also click the down arrow next to the **Evaluate** button on the settings window to view each table. You should get about 0.29 mΩ and 0.11 mH, respectively.

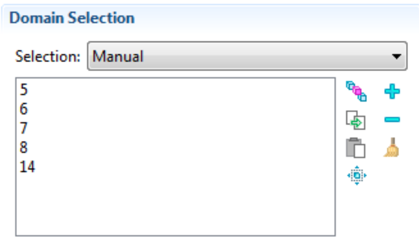


MAGNETIC FIELDS (MF)

As can be seen in the Magnetic Flux Density plot, this is effectively contained within the core. Thus, there should be little need to use infinite elements in this model. Solve the model without the infinite elements to test this hypothesis.

1 In the **Model Builder** under **Model 1**, click **Magnetic Fields** .

2 In the **Magnetic Fields** settings window, select domains 5–8 and 14 only.

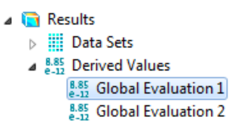


STUDY 1

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

DERIVED VALUES

1 In the **Model Builder** under **Results>Derived Values**, click **Global Evaluation 1** .



2 In the **Global Evaluation** settings window, click the **Evaluate** button. The **Inductance** value displays in the **Results** window table.

3 In the **Model Builder**, click **Global Evaluation 2** .

4 In the **Global Evaluation** settings window, click the **Evaluate** button. The **Resistance** value displays in the **Results** window table. The results change very little.

MODEL WIZARD

Next, add a simple circuit to the model.

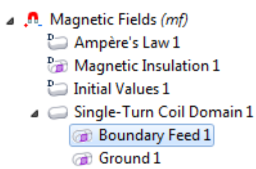
1 In the **Model Builder**, right-click **Model 1** and choose **Add Physics**.

2 In the **Add physics** tree under **AC/DC**, double-click **Electrical Circuit (cir)** to add it to the **Selected physics** list. You do not need to add any studies. Click **Finish**.

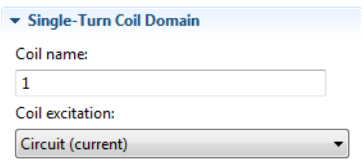
MAGNETIC FIELDS (MF)

Change the field excitation so that it can connect to the circuit part of the model.

1 In the **Model Builder**, under **Magnetic Fields>Single-Turn Coil Domain 1** click **Boundary Feed 1**.



2 In the **Boundary Feed** settings window under **Single-Turn Coil Domain**, choose **Circuit (current)** from the **Coil excitation** list.



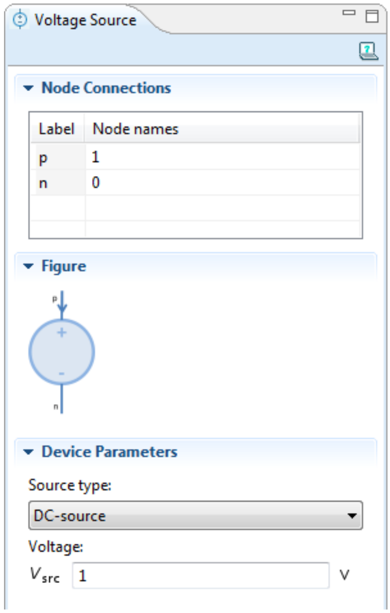
ELECTRICAL CIRCUIT (CIR)

Add a resistor in series with the inductor and drive. This is done for both by applying a voltage source.

Voltage Source 1

1 In the **Model Builder** under **Model 1**, right-click **Electrical Circuit** and choose **Voltage Source**.

The default settings correspond to a DC source of 1 V between nodes 1 and 0 in the circuit net. Keep these settings.

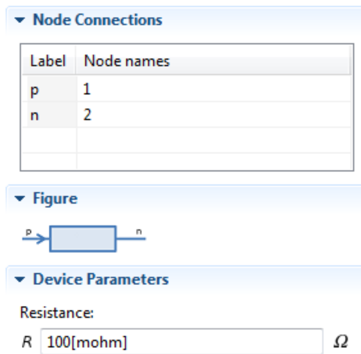


Resistor 1

1 In the **Model Builder**, right-click **Electrical Circuit** and choose **Resistor**.

2 Go to the **Resistor** settings window. Under **Node Connections** enter `1` and `2` in the **Node names** table as in the figure to the right.

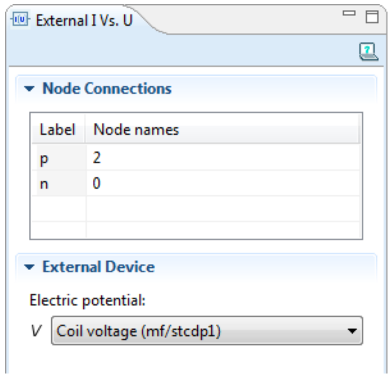
3 Under **Device Parameters** in the R field, enter `100[mohm]`.



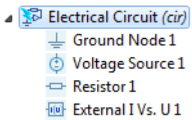
External I Vs. U 1

There is a special feature for connecting the circuit to the finite elements model.

1. In the **Model Builder**, right-click **Electrical Circuit** and choose **External I Vs. U**.
2. In the **External I Vs. U** settings window under **Node Connections**, enter **Node names** as in to the figure to the right.
3. Under **External Device**, from the **Electric potential** V list choose **Coil voltage (mf/stcdp1)**.



The sequence of nodes under **Electrical Circuit** should match this figure.

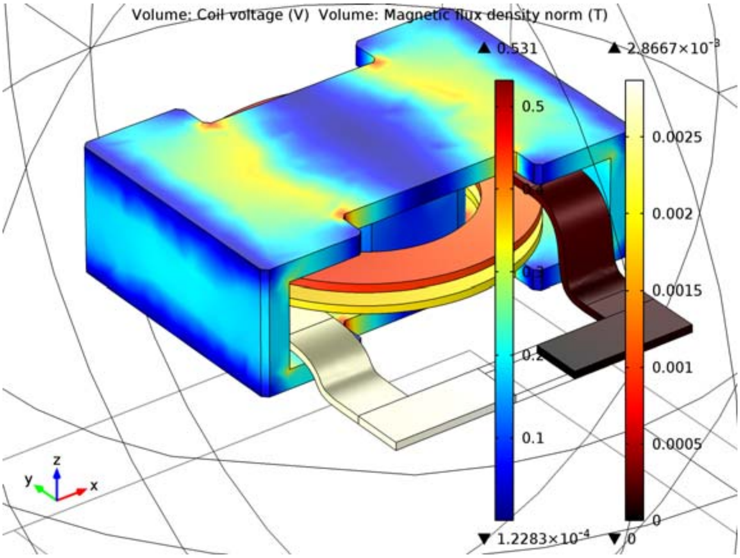


STUDY 1

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

RESULTS

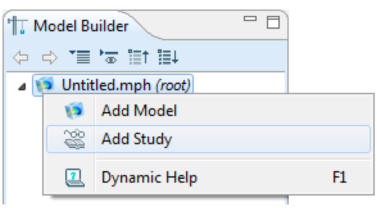
The current is now limited to approximately 10 A by the external resistor which has a much larger resistance than that of the winding.



MODEL WIZARD - ADD A SECOND STUDY

Now, set up the model for computing the frequency-dependent impedance.

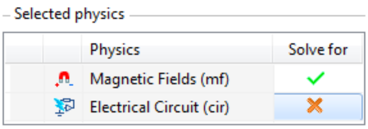
1 In the **Model Builder**, right-click the top node **Untitled.mph (root)** and choose **Add Study**.



2 In the **Select Study Type** window that opens, under **Studies>Preset Studies for Selected Physics** click **Frequency Domain** .

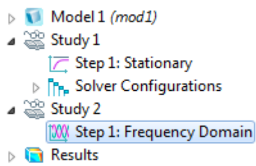
3 Find the **Selected physics** subsection.

4 In the **Selected physics** table under **Solve for**, click to change the check mark to an X and remove the **Electrical Circuit (cir)** interface.



5 Click **Finish**.

A **Study 2** node with a **Frequency Domain** study step is added to the **Model Builder**.



DEFINITIONS

At high frequencies, the skin depth of the copper conductor in the winding becomes difficult to resolve. The solution is to exclude the interior of the copper domain from the simulation and instead represent the winding by a boundary condition, which introduces surface losses via an equivalent surface impedance. For this purpose add a boundary selection and an associated surface material.

Conductor Boundaries

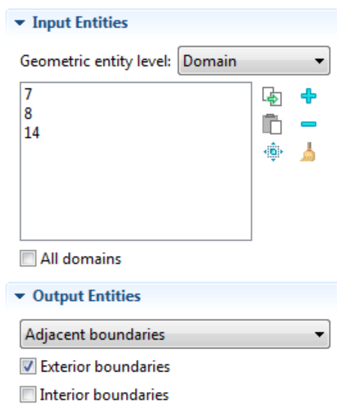
1 In the **Model Builder** under **Model 1**, right-click **Definitions** and choose **Selections>Explicit**.

2 Select Domains 7, 8, and 14 only.

3 Go to the **Explicit** settings window. Under **Output Entities** from the **Output entities** list, select **Adjacent boundaries**.

4 Press F2 and choose **Rename**.

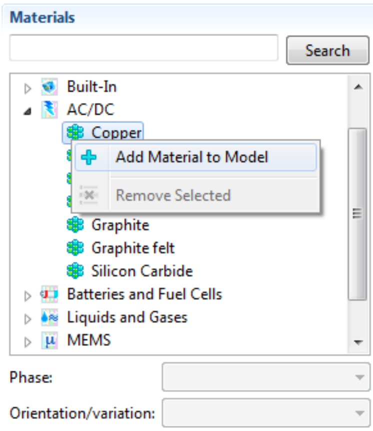
5 In the **Rename Explicit** window enter `Conductor Boundaries` in the **New name** field. Click **OK**.



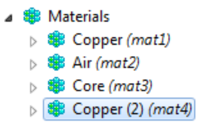
MATERIALS

Copper (2)

1 From the main menu, select **View>Material Browser**. In the **Materials** tree under **AC/DC**, right-click **Copper** and choose **Add Material to Model**.



2 In the **Model Builder**, click **Copper (2)**.

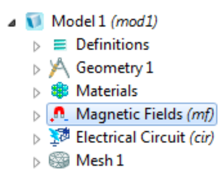


3 Go to the **Material** settings window. Under **Geometric Entity Selection**:

- From the **Geometric entity level** list, select **Boundary**.
- From the **Selection** list, select **Conductor Boundaries**.

MAGNETIC FIELDS (MF)

1 In the **Model Builder** under **Model 1**, click **Magnetic Fields**.



2 Go to the **Magnetic Fields** settings window. Under **Domain Selection** from the **Selection** list, select **Non-conducting without IE**.

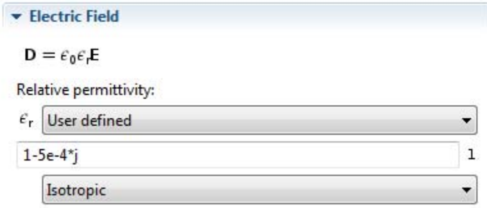
Ampère's Law 2

Apart from the surface loss in the copper conductor, there are also AC losses in the core. The loss in the core is introduced as an effective loss tangent. For this purpose an extra equation/constitutive relation is required.

1 In the **Model Builder**, right-click **Magnetic Fields** and choose **Ampère's Law**. A second **Ampère's Law** node is added to the **Model Builder**.

2 Go to the **Ampère's Law** settings window. Under **Domain Selection** from the **Selection** list, select **Core**.

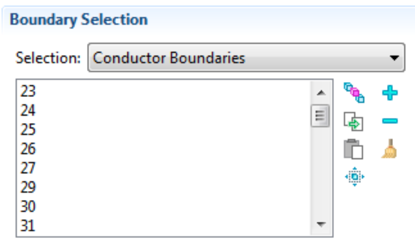
3 Under **Electric Field** from the ε_r list, select **User defined** and enter `1-5e-4*j` in the field.



Impedance Boundary Condition 1

1 In the **Model Builder**, right-click **Magnetic Fields** and at the boundary level, choose **Impedance Boundary Condition**.

2 Go to the **Impedance Boundary Condition** settings window. Under **Boundary Selection** from the **Selection** list, select **Conductor Boundaries**.



Disable the Single-Turn Coil Domain 1

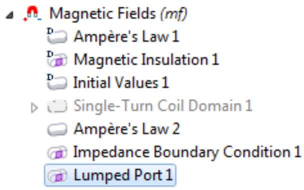
The Single Turn Coil Domain does not apply to an active domain anymore so it needs to be disabled.

- In the **Model Builder**, under **Magnetic Fields** right-click **Single-Turn Coil Domain 1** and choose **Disable**. The node is greyed out.

Lumped Port 1

The electric potential is not available in this physics interface, so to excite the model a different boundary feature, which is more appropriate for high frequency modeling, must be used.

1 In the **Model Builder**, right-click **Magnetic Fields** and at the boundary level choose **Lumped Port**. A **Lumped Port** node is added to the sequence in the **Model Builder**.



2 Go to the **Lumped Port** settings window. Select boundaries 59–62 only.

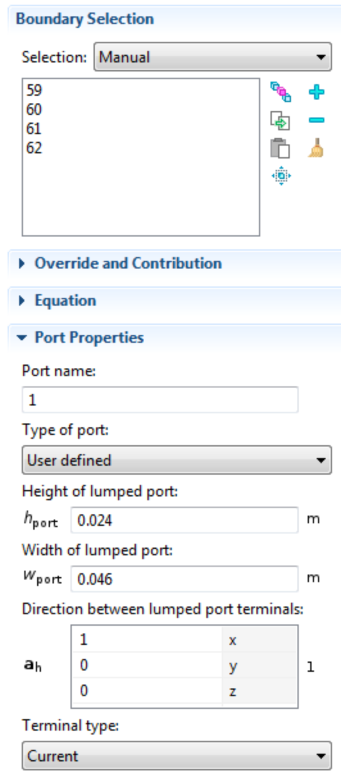
The geometrical parameters of the boundary set need to be entered manually.

3 Under **Port Properties** from the **Type of port** list, select **User defined**.

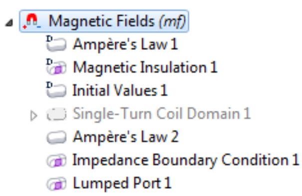
- In the h_{port} field, enter `0.024`.
- In the w_{port} field, enter `0.046`.

4 Specify the $\mathbf{a}_{\mathrm{h}}$ vector as in the figure to the right.

5 From the **Terminal type** list, select **Current**.



The **Model Builder** node sequence under **Magnetic Fields** should match this:



STUDY 2

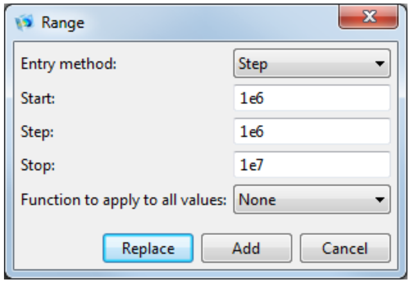
Set up a frequency sweep from 1-10 MHz in steps of 1 MHz.

1 In the **Model Builder**, expand the **Study 2** node and click **Step 1: Frequency Domain**.

2 Under **Study Settings**, click the **Range** button.

3 In the **Range** window, enter these settings:

- In the **Start** field, enter `1e6`.
- In the **Step** field, enter `1e6`.
- In the **Stop** field, enter `1e7`.



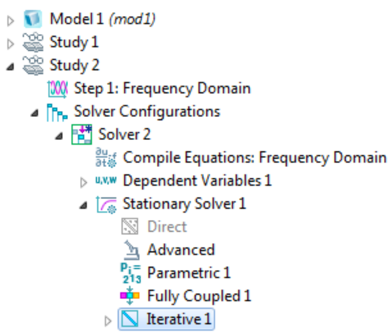
4 Click **Replace**.

Near the resonance frequency, the problem becomes ill-conditioned. If there were no loss, it would even become singular as the field solution then would approach infinity. Thus for a high Q factor, the iterative solver may fail to converge and then a direct solver must be used. Here it is sufficient to tweak the iterative solver to use a more robust preconditioner.

Solver 2

1 In the **Model Builder**, right-click **Study 2** and choose **Show Default Solver**.

2 In the **Model Builder**, expand the **Study 2 >Solver Configurations** nodes as in the figure to the right and then click the **Iterative 1** node.



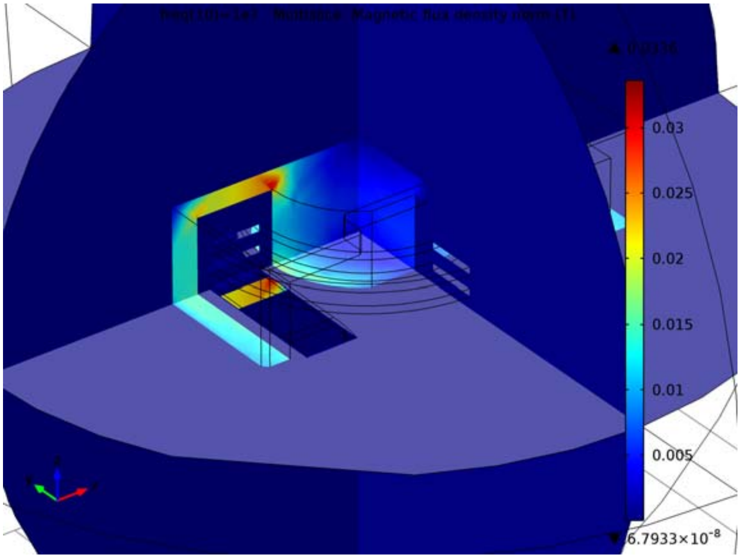
3 Go to the **Iterative** settings window. Under **General** select **Right** from the **Preconditioning** list.

4 In the **Model Builder**, right-click **Study 2** and choose **Compute**.

RESULTS

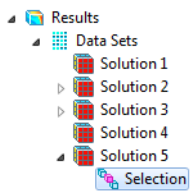
Magnetic Flux Density

Again check the default plot for any modeling errors. Note that the magnetic flux density inside the copper winding is not computed as this domain was excluded.



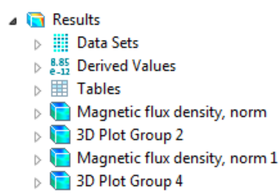
Plot the surface current distribution at the lowest frequency solved for. This frequency is below resonance and the device is still of an inductive nature.

Data Sets

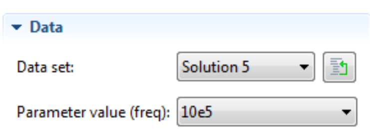


1. In the **Model Builder** under **Results>Data Sets**, right-click **Solution 4** and choose **Duplicate**.
2. Right-click **Solution 5** and choose **Add Selection**.
3. Go to the **Selection** settings window. Under **Geometric Entity Selection** from the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Conductor Boundaries**.

3D Plot Group 4



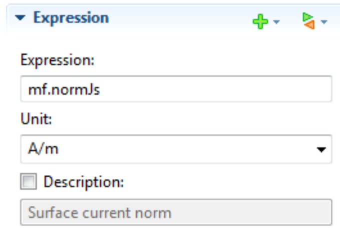
1. In the **Model Builder**, right-click **Results** and choose **3D Plot Group**. A **3D Plot Group 4** node is added to the sequence.
2. Go to the **3D Plot Group** settings window. Under **Data** from the **Data set** list, choose **Solution 5**.



3 From the **Parameter value (freq)** list, choose **10e5**.

4 Right-click **3D Plot Group 4** and choose **Surface**.

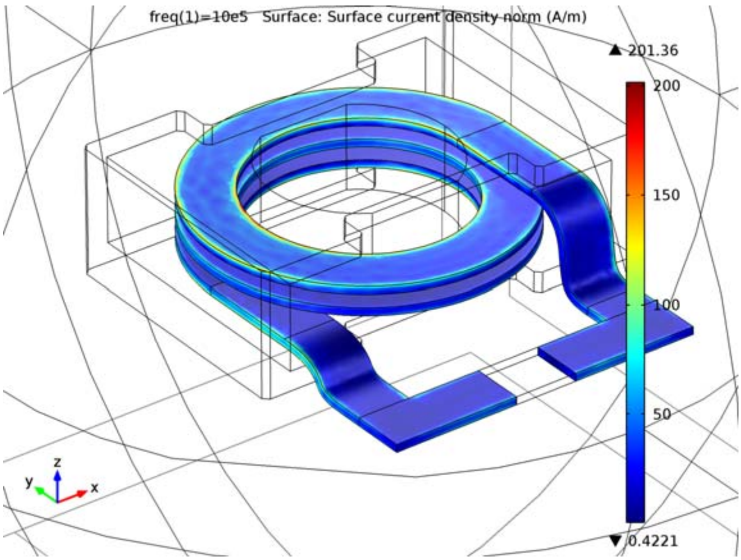
5 Go to the **Surface** settings window. In the upper-right corner of the **Expression** section, click **Replace Expression**.



6 From the menu, choose **Magnetic Fields>Currents and charge>Surface current norm (mf.normJs)** (when you know the variable name, as in this exercise, you can also enter `mf.normJs` in the **Expression** field).

7 Click the **Plot** button.

The plot shows the surface current distribution and should match this figure.



Change to the highest frequency solved for and compare the results. Finish the modeling session by plotting the real and imaginary parts of the coil impedance.

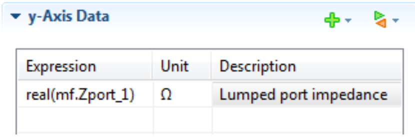
1D Plot Group 5

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

2 In the **1D Plot Group** settings window under **Data**, select **Solution 4** from the **Data set** list.

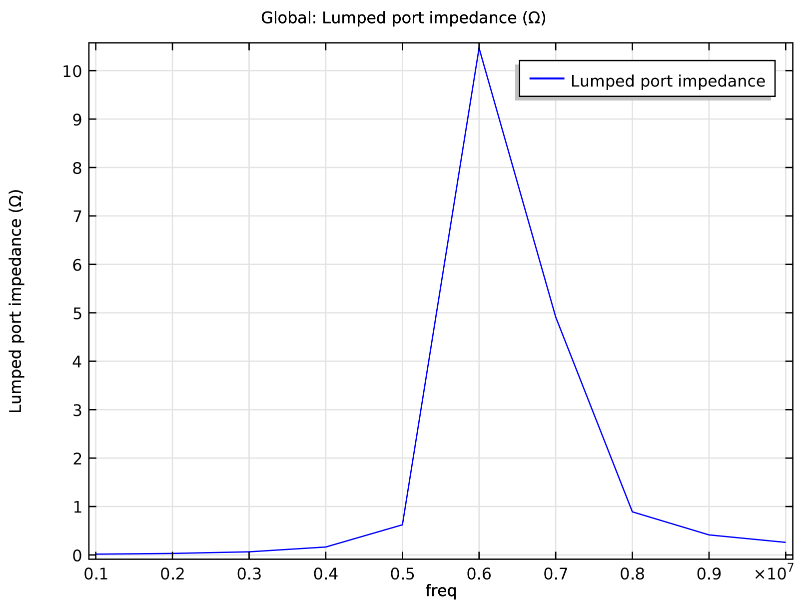
3 Under **Results** right-click **1D Plot Group 5** and choose **Global**.

4 Go to the **Global** settings window. Under **y-Axis Data** in the **Expression** column, enter **real(mf.Zport_1)**. In the **Description** column, enter **Lumped port impedance**.



Note: In general, you can click the **Replace Expression** to see what variables are available in the predefined lists. In this case, you can enter the information directly into the table.

5 Click the **Plot** button.



The resistive part of the coil impedance peaks at the resonance frequency.

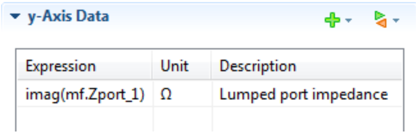
1D Plot Group 6

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

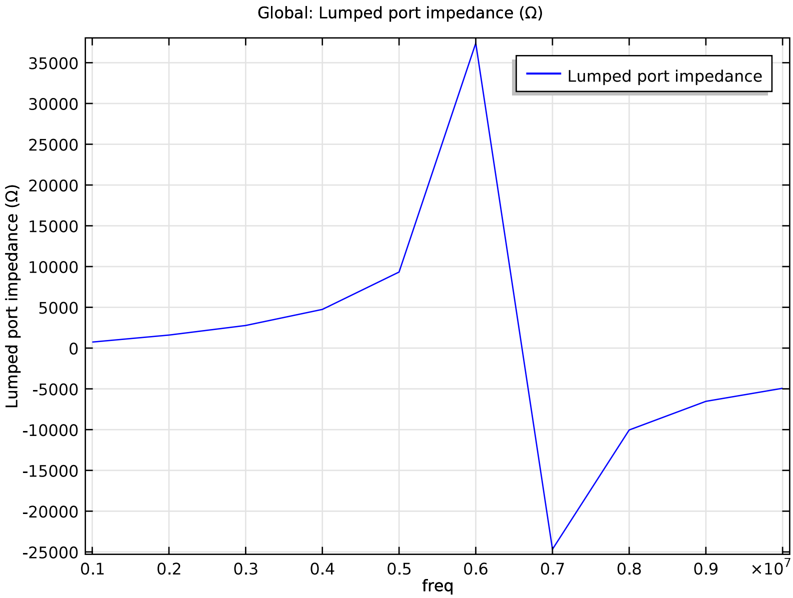
2 Go to the **1D Plot Group** settings window. Under **Data** from the **Data set** list, select **Solution 4**.

3 Under **Results** right-click **1D Plot Group 6** and choose **Global**.

4 Go to the **Global** settings window. Under **y-Axis Data** in the **Expression** column, enter **imag(mf.Zport_1)**. In the **Description** column, enter **Lumped port impedance**.



5 Click the **Plot** button.



The reactive part of the coil impedance changes sign when passing through the resonance frequency, going from inductive to capacitive.

As a final step, pick one of the plots to use as a model thumbnail.

1 In the **Model Builder** under **Results** click **3D Plot Group 2**.

2 From the **File** menu, choose **Save Model Thumbnail**.

To view the thumbnail image, click the **Root** node and look under the **Model Thumbnail** section. Make adjustments to the image in the **Graphics** window using the toolbar buttons until the image is one that is suitable to your purposes.

This concludes this introduction.