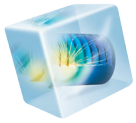


AC/DC Module

User's Guide

VERSION 4.3

COMSOL

AC/DC Module User's Guide

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. CM020101

Contents

Chapter 1: Introduction

Chapter 2: Review of Electromagnetics

Chapter 3: Modeling with the AC/DC Module

Chapter 4: The Electric Field Interfaces

Chapter 5: The Magnetic Field Interfaces

Chapter 6: The Magnetic and Electric Fields Interface

Chapter 7: The Electrical Circuit Interface

Chapter 8: The Heat Transfer Branch

Chapter 9: Materials

Chapter 10: Glossary

1

Introduction

This guide describes the AC/DC Module, an optional add-on package for COMSOL Multiphysics designed to assist you to solve and model low-frequency electromagnetics.

This chapter introduces you to the capabilities of the module including an introduction to the modeling stages and some realistic and illustrative models. A summary of the physics interfaces and where you can find documentation and model examples is also included. The last section is a brief overview with links to each chapter in this guide.

In this chapter:

About the AC/DC Module

In this section:

What Can the AC/DC Module Do?

The AC/DC Module provides a unique environment for simulation of AC/DC electromagnetics in 2D and 3D. The module is a powerful tool for detailed analysis of coils, capacitors, and electrical machinery. With this module you can run static, quasi-static, transient, and time-harmonic simulations in an easy-to-use graphical user interface.

The available physics interfaces cover the following types of electromagnetics field simulations:

- Electrostatics
- Electric currents in conductive media
- Magnetostatics
- Low-frequency electromagnetics

Material properties include inhomogeneous and fully anisotropic materials, media with gains or losses, and complex-valued material properties. Infinite elements makes it possible to model unbounded domains. In addition to the standard results and visualization functionality, the module supports direct computation of lumped parameters such as capacitances and inductances as well as electromagnetic forces.

Like all COMSOL modules, there is a library of ready-to-run models that make it quicker and easier to analyze discipline-specific problems. In addition, any model you develop is described in terms of the underlying partial differential equations, offering a unique way to see the underlying physical laws of a simulation.

The interfaces are fully multiphysics enabled—couple them to any other interface in COMSOL Multiphysics or the other modules. For example, to find the heat distribution in a motor, first find the current in the coils using one of the quasi-static interfaces in this module and then couple it to a heat equation in the main COMSOL Multiphysics package or the Heat Transfer Module. This forms a powerful *multiphysics* model that solves all the equations simultaneously.

COMSOL Multiphysics also has an interface to the MATLAB technical computing environment. If you have a MATLAB license, save it as a Model M-file—a script file that runs in MATLAB. This module also provides interfaces for modeling electrical circuits and importing ECAD drawings.


See Also

- Building a COMSOL Model in the *COMSOL Multiphysics User's Guide*
- AC/DC Module Physics Guide
- AC/DC Module Study Availability
- Where Do I Access the Documentation and Model Library?
- Typographical Conventions

AC/DC Module Physics Guide

The physics interfaces in the AC/DC Module form a complete set of simulation tools for electromagnetic field simulations. To select the right physics interface for describing the real-life physics, the geometric properties and the time variations of the fields need to be considered. The interfaces solve for these physical quantities—the *electric scalar potential* V, the *magnetic vector potential* $\mathbf{A}$, and the *magnetic scalar potential* V_{m}.

Each interface has a *Tag* which is of special importance when performing multiphysics simulations. This tag helps distinguish between physics interfaces and the variables defined by the interface have an underscore plus the physics interface tag appended to their names.

The *Model Wizard* is an easy way to select the physics interface and study type when creating a model for the first time, and physics interfaces can be added to an existing model at any time. Full instructions for selecting interfaces and setting up a model are in the *COMSOL Multiphysics User's Guide*.

In 2D, in-plane and out-of-plane variants are available for problems with a planar symmetry as well as axisymmetric interfaces for problems with a cylindrical symmetry. See

Important	When using an axisymmetric interface it is important that the horizontal axis represents the r direction and the vertical axis the z direction, and that the geometry in the right half-plane (that is, for positive r only) must be created.

What Problems Can You Solve? and Table 1-1 for information about the available study types and variables. See also Overview of the User's Guide for links to the chapters in this guide.

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
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	2D	stationary; time dependent
Electrical Circuit		cir	Not space dependent	stationary; frequency domain; time dependent

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
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.

AC/DC Module Study Availability

TABLE 1-1: AC/DC MODULE DEPENDENT VARIABLES, FIELD COMPONENTS, AND PRESET STUDY AVAILABILITY

PHYSICS INTERFACE	TAG	DEPENDENT VARIABLES	FIELD COMPONENTS *				PRESET STUDIES **					
			MAGNETIC FIELD	ELECTRIC FIELD	MAGNETIC POTENTIAL	CURRENT DENSITY	STATIONARY	TIME DEPENDENT	FREQUENCY DOMAIN	SMALL-SIGNAL ANALYSIS, FREQUENCY DOMAIN	FREQUENCY-STATIONARY	FREQUENCY-TRANSIENT
Electrostatics	es	V		x y z			√	√				
Electric Currents	ec	V		x y z		x y z	√	√	√	√		
Electric Currents, Shell	ecs	V		x y z		x y z	√	√	√	√		
Magnetic Fields	mf	$\mathbf{A}$	x y z	x y z	x y z	x y z	√	√	√	√		
Magnetic and Electric Fields	mef	V, $\mathbf{A}$	x y z	x y z	x y z	x y z	√		√	√		

TABLE 1-1: AC/DC MODULE DEPENDENT VARIABLES, FIELD COMPONENTS, AND PRESET STUDY AVAILABILITY

PHYSICS INTERFACE	TAG	DEPENDENT VARIABLES	FIELD COMPONENTS*				PRESET STUDIES**					
			MAGNETIC FIELD	ELECTRIC FIELD	MAGNETIC POTENTIAL	CURRENT DENSITY	STATIONARY	TIME DEPENDENT	FREQUENCY DOMAIN	SMALL-SIGNAL ANALYSIS, FREQUENCY DOMAIN	FREQUENCY-STATIONARY	FREQUENCY-TRANSIENT
Magnetic Fields, No Currents	mfnc	V_m	x y z				√	√				
Rotating Machinery, Magnetic	rmm	$\mathbf{A}$	x y	z	z	z	√	√				
Electrical Circuit	cir	not applicable					√	√	√			
Induction Heating	ih	$\mathbf{A}$,T, J	x y z	x y z	x y z	x y z	√	√	√		√	√

*These are the nonzero field components. For Cartesian coordinates, these are indexed by x, y, and z; for cylindrical coordinates, r, φ, and z are used.

**Custom studies are also available based on the interface, for example, Eigenfrequency and Eigenvalue.

The Model Builder Show and Hide Physics Options

There are several features available on many physics interfaces or individual nodes. This section is a short overview of the options and includes links to the *COMSOL Multiphysics User's Guide* or *COMSOL Multiphysics Reference Guide* where additional information is available.

Important

The links to the features described in the *COMSOL Multiphysics User's Guide* and *COMSOL Multiphysics Reference Guide* do not work in the PDF, only from within the online help.

 Tip	To locate and search all the documentation for this information, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

SHOW MORE PHYSICS OPTIONS

To display additional features for the physics interfaces and feature nodes, click the **Show** button ( ) on the **Model Builder** and then select the applicable option.

After clicking the **Show** button ( ), some sections display on the settings window when a node is clicked and other features are available from the context menu when a node is right-clicked. For each, the additional sections that can be displayed include **Equation**, **Advanced Settings**, **Discretization**, **Consistent Stabilization**, and **Inconsistent Stabilization**.

You can also click the **Expand Sections** button ( ) in the **Model Builder** to always show some sections or click the **Show** button ( ) and select **Reset to Default** to reset to display only the **Equation** and **Override and Contribution** sections.

For most physics nodes, both the **Equation** and **Override and Contribution** sections are always available. Click the **Show** button ( ) and then select **Equation View** to display the **Equation View** node under all physics nodes in the **Model Builder**.

Availability of each feature, and whether it is described for a particular physics node, is based on the individual physics selected. For example, the **Discretization**, **Advanced Settings**, **Consistent Stabilization**, and **Inconsistent Stabilization** sections are often described individually throughout the documentation as there are unique settings.

SECTION	CROSS REFERENCE	LOCATION IN COMSOL MULTIPHYSICS USER GUIDE OR REFERENCE GUIDE
Show More Options and Expand Sections	• Showing and Expanding Advanced Physics Sections • The Model Builder Window	User's Guide
Discretization	• Show Discretization • Element Types and Discretization	User's Guide
	• Finite Elements • Discretization of the Equations	Reference Guide

SECTION	CROSS REFERENCE	LOCATION IN COMSOL MULTIPHYSICS USER GUIDE OR REFERENCE GUIDE
Discretization - Splitting of complex variables	Compile Equations	Reference Guide
Pair Selection	• Identity and Contact Pairs • Specifying Boundary Conditions for Identity Pairs	User's Guide
Consistent and Inconsistent Stabilization	Show Stabilization	User's Guide
	• Stabilization Techniques • Numerical Stabilization	Reference Guide
Geometry	Working with Geometry	User's Guide
Constraint Settings	Using Weak Constraints	User's Guide

Where Do I Access the Documentation and Model Library?

A number of Internet resources provide more information about COMSOL Multiphysics, including licensing and technical information. The electronic documentation, Dynamic Help, and the Model Library are all accessed through the COMSOL Desktop.

Important

If you are reading the documentation as a PDF file on your computer, the blue links do not work to open a model or content referenced in a different user's guide. However, if you are using the online help in COMSOL Multiphysics, these links work to other modules, model examples, and documentation sets.

THE DOCUMENTATION

The *COMSOL Multiphysics User's Guide* and *COMSOL Multiphysics Reference Guide* describe all interfaces and functionality included with the basic COMSOL Multiphysics license. These guides also have instructions about how to use COMSOL Multiphysics and how to access the documentation electronically through the COMSOL Multiphysics help desk.

To locate and search all the documentation, in COMSOL Multiphysics:

- Press F1 for Dynamic Help,
- Click the buttons on the toolbar, or
- Select **Help>Documentation** ( ) or **Help>Dynamic Help** ( ) from the main menu

and then either enter a search term or look under a specific module in the documentation tree.

THE MODEL LIBRARY

Each model comes with documentation that includes a theoretical background and step-by-step instructions to create the model. The models are available in COMSOL as MPH-files that you can open for further investigation. You can use the step-by-step instructions and the actual models as a template for your own modeling and applications.

SI units are used to describe the relevant properties, parameters, and dimensions in most examples, but other unit systems are available.

To open the Model Library, select **View>Model Library** ( ) from the main menu, and then search by model name or browse under a module folder name. Click to highlight any model of interest, and select **Open Model and PDF** to open both the model and the documentation explaining how to build the model. Alternatively, click the **Dynamic Help** button ( ) or select **Help>Documentation** in COMSOL to search by name or browse by module.

The model libraries are updated on a regular basis by COMSOL in order to add new models and to improve existing models. Choose **View>Model Library Update** ( ) to update your model library to include the latest versions of the model examples.

If you have any feedback or suggestions for additional models for the library (including those developed by you), feel free to contact us at info@comsol.com.

CONTACTING COMSOL BY EMAIL

For general product information, contact COMSOL at info@comsol.com.

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and case number is sent to you by email.

COMSOL WEB SITES

Main Corporate web site	www.comsol.com
Worldwide contact information	www.comsol.com/contact
Technical Support main page	www.comsol.com/support
Support Knowledge Base	www.comsol.com/support/knowledgebase
Product updates	www.comsol.com/support/updates
COMSOL User Community	www.comsol.com/community

Typographical Conventions

All COMSOL user's guides use a set of consistent typographical conventions that make it easier to follow the discussion, understand what you can expect to see on the graphical user interface (GUI), and know which data must be entered into various data-entry fields.

In particular, these conventions are used throughout the documentation:

CONVENTION	EXAMPLE
text highlighted in blue	Click text highlighted in blue to go to other information in the PDF. When you are using the online help desk in COMSOL Multiphysics, these links also work to other modules, model examples, and documentation sets.
boldface font	A **boldface** font indicates that the given word(s) appear exactly that way on the COMSOL Desktop (or, for toolbar buttons, in the corresponding tip). For example, the **Model Builder** window () is often referred to and this is the window that contains the model tree. As another example, the instructions might say to click the **Zoom Extents** button (), and this means that when you hover over the button with your mouse, the same label displays on the COMSOL Desktop.
Forward arrow symbol **>**	The forward arrow symbol **>** is instructing you to select a series of menu items in a specific order. For example, **Options>Preferences** is equivalent to: From the **Options** menu, choose **Preferences**.
`Code` (monospace) font	A `Code` (monospace) font indicates you are to make a keyboard entry in the user interface. You might see an instruction such as "Enter (or type) `1.25` in the **Current density** field." The monospace font also is an indication of programming code or a variable name.

CONVENTION	EXAMPLE
Italic *Code* (monospace) font	An italic *Code* (monospace) font indicates user inputs and parts of names that can vary or be defined by the user.
Arrow brackets <> following the Code (monospace) or *Code* (italic) fonts	The arrow brackets included in round brackets after either a monospace Code or an italic *Code* font means that the content in the string can be freely chosen or entered by the user, such as feature tags. For example, `model.geom(<tag>)` where <*tag*> is the geometry's tag (an identifier of your choice). When the string is predefined by COMSOL, no bracket is used and this indicates that this is a finite set, such as a feature name.

KEY TO THE GRAPHICS

Throughout the documentation, additional icons are used to help navigate the information. These categories are used to draw your attention to the information based on the level of importance, although it is always recommended that you read these text boxes.

ICON	NAME	DESCRIPTION
	Caution	A Caution icon is used to indicate that the user should proceed carefully and consider the next steps. It might mean that an action is required, or if the instructions are not followed, that there will be problems with the model solution.
	Important	An Important icon is used to indicate that the information provided is key to the model building, design, or solution. The information is of higher importance than a note or tip, and the user should endeavor to follow the instructions.
	Note	A Note icon is used to indicate that the information may be of use to the user. It is recommended that the user read the text.
	Tip	A Tip icon is used to provide information, reminders, short cuts, suggestions of how to improve model design, and other information that may or may not be useful to the user.
	See Also	The See Also icon indicates that other useful information is located in the named section. If you are working on line, click the hyperlink to go to the information directly. When the link is outside of the current PDF document, the text indicates this, for example See The Laminar Flow Interface in the *COMSOL Multiphysics User's Guide*. Note that if you are in COMSOL Multiphysics' online help, the link will work.

ICON	NAME	DESCRIPTION
	Model	The Model icon is used in the documentation as well as in COMSOL Multiphysics from the View>Model Library menu. If you are working online, click the link to go to the PDF version of the step-by-step instructions. In some cases, a model is only available if you have a license for a specific module. These examples occur in the COMSOL Multiphysics User's Guide. The Model Library path describes how to find the actual model in COMSOL Multiphysics, for example If you have the RF Module, see Radar Cross Section: Model Library path **RF_Module/Tutorial_Models/radar_cross_section**
Space Dimension		Another set of icons are also used in the Model Builder—the model space dimension is indicated by 0D , 1D , 1D axial symmetry  , 2D  , 2D axial symmetry  , and 3D  icons. These icons are also used in the documentation to clearly list the differences to an interface, feature node, or theory section, which are based on space dimension.

Overview of the User's Guide

The *AC/DC Module User's Guide* gets you started with modeling using COMSOL Multiphysics. The information in this guide is specific to this module. Instructions how to use COMSOL in general are included with the *COMSOL Multiphysics User's Guide*.


Tip

As detailed in the section Where Do I Access the Documentation and Model Library? this information is also searchable from the COMSOL Multiphysics software **Help** menu.

TABLE OF CONTENTS, GLOSSARY, AND INDEX

To help you navigate through this guide, see the Contents, Glossary, and Index.

THEORY OF ELECTROMAGNETICS

In the Review of Electromagnetics chapter contains an overview of the theory behind the AC/DC Module. It is intended for readers that wish to understand what goes on in the background when using the physics interfaces and discusses the Fundamentals of Electromagnetics, Electromagnetic Forces, and Electromagnetic Quantities.

MODELING WITH THE AC/DC MODULE

In the Modeling with the AC/DC Module chapter, the goal is to familiarize you with the modeling procedure using this particular module. Topics include Preparing for Modeling, Infinite Elements, Force and Torque Computations, Lumped Parameters, and Importing ECAD Files.

ELECTRIC FIELDS

The Electric Field Interfaces chapter describes these interfaces and includes the underlying theory for each interface at the end of the chapter:

- The Electrostatics Interface, which simulates electric fields in dielectric materials with a fixed charge present. Preset stationary and time dependent study types are available.

- The Electric Currents Interface, which simulates the current in a conductive and capacitive material under the influence of an electric field. All three study types (stationary, frequency domain, and time dependent) are available.
- The Electric Currents, Shell Interface, which simulates the current in a conductive and capacitive shell under the influence of an electric field. All three study types (stationary, frequency domain and time dependent) are available.

MAGNETIC FIELDS

The Magnetic Field Interfaces chapter describes these interfaces and includes the underlying theory for each interface at the end of the chapter:

- The Magnetic Fields Interface, which handles problems for magnetic fields with prescribed currents. All three study types (stationary, frequency domain, and time dependent) are available.
- The Magnetic Fields, No Currents Interface, which handles magnetic fields without currents. When no currents are present, the problem is easier to solve using the magnetic scalar potential. The stationary and time dependent study types are available.
- The Rotating Machinery, Magnetic Interface is available with 2D models only. It combines an out-of-plane magnetic fields (magnetic vector potential) formulation 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.

MAGNETIC AND ELECTRIC FIELDS

The Magnetic and Electric Fields Interface chapter describes the interface, which handles problems for magnetic and electric fields. It is based on the magnetic vector potential and the electric scalar potential. The stationary and frequency domain study types are available. The underlying theory for the interface is included at the end of the chapter.

ELECTRICAL CIRCUIT

The Electrical Circuit Interface chapter describes the interface, which has the equations for modeling electrical circuits with or without connections to a distributed fields model, solving for the voltages, currents, and charges associated with the circuit elements. The underlying theory for the interface is included at the end of the chapter.

HEAT TRANSFER

The Heat Transfer Branch chapter describes the interface, which combines all features from the Magnetic Fields interface in the time harmonic formulation with the Heat Transfer interface for modeling of induction and eddy current heating.

Heat transfer through conduction and convection in solids and free media (fluids) is supported by physics interfaces shipped with the basic COMSOL Multiphysics license.


See Also

The Heat Transfer Interface, The Joule Heating Interface, and Theory for the Heat Transfer Interfaces in the *COMSOL Multiphysics User's Guide*.

MATERIALS

The Materials chapter has details about the electromagnetic material properties that you can store in the material databases such as electrical conductivity and resistivity, relative permittivity, relative permeability, nonlinear BH-curves, and refractive index.

2

Review of Electromagnetics

This chapter contains an overview of the theory behind the AC/DC Module. It is intended for readers that wish to understand what goes on in the background when using the physics interfaces.

In this chapter:

Fundamentals of Electromagnetics

In this section:

Maxwell's Equations

The problem of electromagnetic analysis on a macroscopic level is that of solving *Maxwell's equations* subject to certain boundary conditions. Maxwell's equations are a set of equations, written in differential or integral form, stating the relationships between the fundamental electromagnetic quantities. These quantities are:

- Electric field intensity **E**
- Electric displacement or electric flux density **D**
- Magnetic field intensity **H**
- Magnetic flux density **B**
- Current density **J**
- Electric charge density ρ

The equations can be formulated in differential form or integral form. The differential form is presented here because it leads to differential equations that the finite element method can handle. For general time-varying fields, Maxwell's equations can be written as

$$\nabla \times \mathbf{H} = \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t}$$

$$\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t}$$

$$\nabla \cdot \mathbf{D} = \rho$$

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

The first two equations are also referred to as *Maxwell-Ampère's law* and *Faraday's law*, respectively. Equation three and four are two forms of *Gauss' law*: the electric and magnetic form, respectively.

Another fundamental equation is the *equation of continuity*

$$\nabla \cdot \mathbf{J} = -\frac{\partial \rho}{\partial t}$$

Out of the five equations mentioned, only three are independent. The first two combined with either the electric form of Gauss' law or the equation of continuity form such an independent system.

Constitutive Relations

To obtain a closed system, the equations include *constitutive relations* that describe the macroscopic properties of the medium. They are given as

$$\begin{aligned} \mathbf{D} &= \varepsilon_0 \mathbf{E} + \mathbf{P} \\ \mathbf{B} &= \mu_0(\mathbf{H} + \mathbf{M}) \\ \mathbf{J} &= \sigma \mathbf{E} \end{aligned} \tag{2-1}$$

where ε_0 is the *permittivity of vacuum*, μ_0 is the *permeability of vacuum*, and σ the *electrical conductivity*. In the SI system, the permeability of vacuum is chosen to be $4\pi \cdot 10^{-7}$ H/m. The velocity of an electromagnetic wave in vacuum is given as c_0 and the permittivity of vacuum is derived from the relation

$$\varepsilon_0 = \frac{1}{c_0^2 \mu_0} = 8.854 \cdot 10^{-12} \text{ F/m} \approx \frac{1}{36\pi} \cdot 10^{-9} \text{ F/m}$$

The electromagnetic constants ε_0, μ_0, and c_0 are available in COMSOL Multiphysics as predefined *physical constants*.

The *electric polarization vector* $\mathbf{P}$ describes how the material is polarized when an electric field $\mathbf{E}$ is present. It can be interpreted as the volume density of electric dipole

moments. $\mathbf{P}$ is generally a function of $\mathbf{E}$. Some materials can have a nonzero $\mathbf{P}$ also when there is no electric field present.

The *magnetization vector* $\mathbf{M}$ similarly describes how the material is magnetized when a magnetic field $\mathbf{H}$ is present. It can be interpreted as the volume density of magnetic dipole moments. $\mathbf{M}$ is generally a function of $\mathbf{H}$. Permanent magnets, for instance, have a nonzero $\mathbf{M}$ also when there is no magnetic field present.

For linear materials, the polarization is directly proportional to the electric field, $\mathbf{P} = \varepsilon_0 \chi_e \mathbf{E}$, where χ_e is the *electric susceptibility*. Similarly in linear materials, the magnetization is directly proportional to the magnetic field, $\mathbf{M} = \chi_m \mathbf{H}$, where χ_m is the *magnetic susceptibility*. For such materials, the constitutive relations are

$$\mathbf{D} = \varepsilon_0(1 + \chi_e)\mathbf{E} = \varepsilon_0 \varepsilon_r \mathbf{E} = \varepsilon \mathbf{E}$$
$$\mathbf{B} = \mu_0(1 + \chi_m)\mathbf{H} = \mu_0 \mu_r \mathbf{H} = \mu \mathbf{H}$$

The parameter ε_r is the *relative permittivity* and μ_r is the *relative permeability* of the material. Usually these are scalar properties but can, in the general case, be 3-by-3 tensors when the material is anisotropic. The properties ε and μ (without subscripts) are the *permittivity* and *permeability* of the material.

GENERALIZED CONSTITUTIVE RELATIONS


See Also

The Charge Conservation feature describes the macroscopic properties of the medium (relating the electric displacement $\mathbf{D}$ with the electric field $\mathbf{E}$) and the applicable material properties.

For nonlinear materials, a generalized form of the constitutive relationships is useful. The relationship used for electric fields is $\mathbf{D} = \varepsilon_0 \varepsilon_r \mathbf{E} + \mathbf{D}_r$ where $\mathbf{D}_r$ is the *remanent displacement*, which is the displacement when no electric field is present.

Similarly, a generalized form of the constitutive relation for the magnetic field is

$$\mathbf{B} = \mu_0 \mu_r \mathbf{H} + \mathbf{B}_r$$

where $\mathbf{B}_r$ is the *remanent magnetic flux density*, which is the magnetic flux density when no magnetic field is present.

For some materials, there is a nonlinear relationship between $\mathbf{B}$ and $\mathbf{H}$ such that

$$\mathbf{B} = f(|\mathbf{H}|)$$

The relation defining the current density is generalized by introducing an externally generated current $\mathbf{J}_e$. The resulting constitutive relation is $\mathbf{J} = \sigma\mathbf{E} + \mathbf{J}_e$.

Potentials

Under certain circumstances it can be helpful to formulate the problems in terms of the *electric scalar potential* V and the *magnetic vector potential* $\mathbf{A}$. They are given by the equalities

$$\mathbf{B} = \nabla \times \mathbf{A}$$
$$\mathbf{E} = -\nabla V - \frac{\partial \mathbf{A}}{\partial t}$$

The defining equation for the magnetic vector potential is a direct consequence of the magnetic Gauss' law. The electric potential results from Faraday's law. In the magnetostatic case where there are no currents present, Maxwell-Ampère's law reduces to $\nabla \times \mathbf{H} = \mathbf{0}$. When this holds, it is also possible to define a *magnetic scalar potential* by the relation $\mathbf{H} = -\nabla V_m$.

Reduced Potential PDE Formulations

The *reduced potential* option introduces the substitution $\mathbf{A} = \mathbf{A}_{red} + \mathbf{A}_{ext}$ into Maxwell-Ampère's law:

$$\nabla\times(\mu^{-1}\nabla\times\mathbf{A}) = \mathbf{J} + \frac{d\mathbf{D}}{dt}$$

DOMAIN EQUATIONS

Time-Harmonic

For time-harmonic quasi-static systems solving for an A formulation, the reduced potential formulation results in the following PDE:

$$(j\omega\sigma - \omega^2\varepsilon)(\mathbf{A}_{ext} + \mathbf{A}_{red}) + \nabla\times(\mu^{-1}\nabla\times(\mathbf{A}_{ext} + \mathbf{A}_{red})) = \mathbf{J}_e$$

Here it is possible to interpret the term $\nabla \times \mathbf{A}_{ext}$ as an additional remanent magnetic flux density and the term $(j\omega\sigma - \omega^2\varepsilon)\mathbf{A}_{ext}$ as an additional external current source.

Transient

Similarly to the time-harmonic formulation, in the transient formulation, the above substitution results in the reduced equation

$$\sigma\frac{\partial}{\partial t}(\mathbf{A}_{\text{ext}} + \mathbf{A}_{\text{red}}) + \nabla\times(\mu^{-1}\nabla\times(\mathbf{A}_{\text{ext}} + \mathbf{A}_{\text{red}})) = \mathbf{J}_{\text{e}}$$

Static

In static formulations, the induced current is zero. Maxwell-Ampère's law re*duces to*

$$\nabla\times(\mu^{-1}\nabla\times(\mathbf{A}_{\text{ext}} + \mathbf{A}_{\text{red}})) = \mathbf{J}_{\text{e}}$$

In this case it is also possible to express the external field through a known external magnetic flux density, $\mathbf{B}_{\text{ext}}$. The domain equation in reduced form then reads

$$\nabla\times(\mu^{-1}(\nabla\times\mathbf{A}_{\text{red}} + \mathbf{B}_{\text{ext}})) = \mathbf{J}_{e}$$

Electromagnetic Energy

The electric and magnetic energies are defined as

$$W_{\text{e}} = \int_V\left(\int_0^D \mathbf{E}\cdot d\mathbf{D}\right)dV = \int_V\left(\int_0^T \mathbf{E}\cdot\frac{\partial\mathbf{D}}{\partial t}dt\right)dV$$
$$W_{\text{m}} = \int_V\left(\int_0^B \mathbf{H}\cdot d\mathbf{B}\right)dV = \int_V\left(\int_0^T \mathbf{H}\cdot\frac{\partial\mathbf{B}}{\partial t}dt\right)dV$$

The time derivatives of these expressions are the electric and magnetic power

$$P_{\text{e}} = \int_V \mathbf{E}\cdot\frac{\partial\mathbf{D}}{\partial t}dV$$
$$P_{\text{m}} = \int_V \mathbf{H}\cdot\frac{\partial\mathbf{B}}{\partial t}dV$$

These quantities are related to the resistive and radiative energy, or energy loss, through Poynting's theorem (Ref. 1)

$$-\int_V\left(\mathbf{E}\cdot\frac{\partial\mathbf{D}}{\partial t} + \mathbf{H}\cdot\frac{\partial\mathbf{B}}{\partial t}\right)dV = \int_V \mathbf{J}\cdot\mathbf{E}dV + \oint_S(\mathbf{E}\times\mathbf{H})\cdot\mathbf{n}dS$$

where V is the computation domain and S is the closed boundary of V.

The first term on the right-hand side represents the resistive losses,

$$P_{\text{h}} = \int_V \mathbf{J}\cdot\mathbf{E}dV$$

which result in heat dissipation in the material. (The current density $\mathbf{J}$ in this expression is the one appearing in Maxwell-Ampère's law.)

The second term on the right-hand side of Poynting's theorem represents the radiative losses,

$$P_{\mathrm{r}} = \oint_S (\mathbf{E} \times \mathbf{H}) \cdot \mathbf{n} dS$$

The quantity $\mathbf{S} = \mathbf{E} \times \mathbf{H}$ is called the Poynting vector.

Under the assumption the material is linear and isotropic, it holds that

$$\mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} = \varepsilon \mathbf{E} \cdot \frac{\partial \mathbf{E}}{\partial t} = \frac{\partial}{\partial t}\left(\frac{1}{2}\varepsilon \mathbf{E} \cdot \mathbf{E}\right)$$

$$\mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} = \frac{1}{\mu}\mathbf{B} \cdot \frac{\partial \mathbf{B}}{\partial t} = \frac{\partial}{\partial t}\left(\frac{1}{2\mu}\mathbf{B} \cdot \mathbf{B}\right)$$

By interchanging the order of differentiation and integration (justified by the fact that the volume is constant and the assumption that the fields are continuous in time), the result is

$$-\frac{\partial}{\partial t}\int_V \left(\frac{1}{2}\varepsilon \mathbf{E} \cdot \mathbf{E} + \frac{1}{2\mu}\mathbf{B} \cdot \mathbf{B}\right) dV = \int_V \mathbf{J} \cdot \mathbf{E} dV + \oint_S (\mathbf{E} \times \mathbf{H}) \cdot \mathbf{n} dS$$

The integrand of the left-hand side is the total electromagnetic energy density

$$w = w_{\mathrm{e}} + w_{\mathrm{m}} = \frac{1}{2}\varepsilon \mathbf{E} \cdot \mathbf{E} + \frac{1}{2\mu}\mathbf{B} \cdot \mathbf{B}$$

The Quasi-Static Approximation and the Lorentz Term

A consequence of Maxwell's equations is that changes in time of currents and charges are not synchronized with changes of the electromagnetic fields. The changes of the fields are always delayed relative to the changes of the sources, reflecting the finite speed of propagation of electromagnetic waves. Under the assumption that this effect can be ignored, it is possible to obtain the electromagnetic fields by considering stationary currents at every instant. This is called the *quasi-static approximation*. The approximation is valid provided that the variations in time are small and that the studied geometries are considerably smaller than the wavelength (Ref. 5).

The quasi-static approximation implies that the equation of continuity can be written as $\nabla \cdot \mathbf{J} = 0$ and that the time derivative of the electric displacement $\partial \mathbf{D}/\partial t$ can be disregarded in Maxwell-Ampère's law.

There are also effects of the motion of the geometries. Consider a geometry moving with velocity $\mathbf{v}$ relative to the reference system. The force per unit charge, $\mathbf{F}/q$, is then given by the *Lorentz force equation*:

$$\frac{\mathbf{F}}{q} = \mathbf{E} + \mathbf{v} \times \mathbf{B}$$

This means that to an observer traveling with the geometry, the force on a charged particle can be interpreted as caused by an electric field $\mathbf{E}' = \mathbf{E} + \mathbf{v} \times \mathbf{B}$. In a conductive medium, the observer accordingly sees the current density

$$\mathbf{J} = \sigma(\mathbf{E} + \mathbf{v} \times \mathbf{B}) + \mathbf{J}_{\mathrm{e}}$$

where $\mathbf{J}_{\mathrm{e}}$ is an externally generated current density.

Maxwell-Ampère's law for quasi-static systems is consequently extended to

$$\nabla \times \mathbf{H} = \sigma(\mathbf{E} + \mathbf{v} \times \mathbf{B}) + \mathbf{J}^{\mathrm{e}}$$

whereas Faraday's law remains unchanged.

Material Properties

Until now, there has only been a formal introduction of the constitutive relations. These seemingly simple relations can be quite complicated at times. There are four main groups of materials where they require some consideration. A given material can belong to one or more of these groups. The groups are:

- Inhomogeneous materials
- Anisotropic materials
- Nonlinear materials
- Dispersive materials

A material can belong to one or more of these groups.

INHOMOGENEOUS MATERIALS

Inhomogeneous materials are the least complicated. An inhomogeneous medium is one in which the constitutive parameters vary with the space coordinates so that different field properties prevail at different parts of the material structure.

ANISOTROPIC MATERIALS

For anisotropic materials the field relationships at any point differ for different directions of propagation. This means that a 3-by-3 tensor is necessary to properly define the constitutive relationships. If this tensor is symmetric, the material is often referred to as *reciprocal*. In such cases, rotate the coordinate system such that a diagonal matrix results. If two of the diagonal entries are equal, the material is *uniaxially anisotropic*. If none of the elements has the same value, the material is *biaxially anisotropic* (Ref. 2). Anisotropic parameters are needed, for example, to examine permittivity in crystals (Ref. 2) and when working with conductivity in solenoids.

NONLINEAR MATERIALS

Nonlinearity is the effect of variations in permittivity or permeability with the intensity of the electromagnetic field. Nonlinearity also includes hysteresis effects, where not only the current field intensities influence the physical properties of the material, but also the history of the field distribution.

DISPERSIVE MATERIALS

Dispersion describes changes in a wave's velocity with wavelength. In the frequency domain dispersion is expressed with a frequency dependence of the constitutive laws.

About the Boundary and Interface Conditions

To get a full description of an electromagnetics problem, boundary conditions must be specified at material interfaces and physical boundaries. At interfaces between two media, the boundary conditions can be expressed mathematically as

$$\mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) = \mathbf{0}$$
$$\mathbf{n}_2 \cdot (\mathbf{D}_1 - \mathbf{D}_2) = \rho_s$$
$$\mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{J}_s$$
$$\mathbf{n}_2 \cdot (\mathbf{B}_1 - \mathbf{B}_2) = 0$$

where ρ_s and $\mathbf{J}_s$ denote *surface charge density* and *surface current density*, respectively, and $\mathbf{n}_2$ is the outward normal from medium 2. Of these four conditions, only two are independent. This is an overdetermined system of equations, so it needs to be reduced. First select either equation one or equation four. Then select either equation two or equation three. Together these selections form a set of two independent conditions.

From these relationships, the interface condition is derived for the current density,

$$\mathbf{n}_2 \cdot (\mathbf{J}_1 - \mathbf{J}_2) = -\frac{\partial \rho_s}{\partial t}$$

INTERFACE BETWEEN A DIELECTRIC AND A PERFECT CONDUCTOR

A perfect conductor has infinite electrical conductivity and thus no internal electric field. Otherwise, it would produce an infinite current density according to the third fundamental constitutive relation. At an interface between a dielectric and a perfect conductor, the boundary conditions for the **E** and **D** fields are simplified. Assume that subscript 1 corresponds to a perfect conductor; then $\mathbf{D}_1 = \mathbf{0}$ and $\mathbf{E}_1 = \mathbf{0}$ in the relationships just given. If, in addition, it is a time-varying case, then $\mathbf{B}_1 = \mathbf{0}$ and $\mathbf{H}_1 = \mathbf{0}$, as well, as a consequence of Maxwell's equations. The result is the following set of boundary conditions for the fields in the dielectric medium for the time-varying case:

$$\begin{aligned} -\mathbf{n}_2 \times \mathbf{E}_2 &= 0 \\ -\mathbf{n}_2 \times \mathbf{H}_2 &= \mathbf{J}_s \\ -\mathbf{n}_2 \cdot \mathbf{D}_2 &= \rho_s \\ -\mathbf{n}_2 \cdot \mathbf{B}_2 &= 0 \end{aligned}$$

Phasors

Whenever a problem is time-harmonic the fields can be written in the form

$$E(\mathbf{r}, t) = \hat{E}(\mathbf{r}) \cos(\omega t + \phi)$$

Instead of using a cosine function for the time dependence, it is more convenient to use an exponential function, by writing the field as

$$E(\mathbf{r}, t) = \hat{E}(\mathbf{r}) \cos(\omega t + \phi) = \mathrm{Re}(\hat{E}(\mathbf{r}) e^{j\phi} e^{j\omega t}) = \mathrm{Re}(\tilde{E}(\mathbf{r}) e^{j\omega t})$$

The field $\tilde{\boldsymbol{E}}(\mathbf{r})$ is a *phasor*, which contains amplitude and phase information of the field but is independent of t. One thing that makes the use of phasors suitable is that a time derivative corresponds to a multiplication by $j\omega$,

$$\frac{\partial \boldsymbol{E}}{\partial t} = \mathrm{Re}(j\omega \tilde{\boldsymbol{E}}(\mathbf{r}) e^{j\omega t})$$

This means that an equation for the phasor can be derived from a time-dependent equation by replacing the time derivatives by a factor $j\omega$. All time-harmonic equations in the AC/DC Module are expressed as equations for the phasors. (The tilde is dropped from the variable denoting the phasor.)

When analyzing the solution of a time-harmonic equation, it is important to remember that the field that has been calculated is a phasor and not a physical field.

For example, all plot functions visualize $\mathrm{Re}(\tilde{\boldsymbol{E}}(\mathbf{r}))$ by default, which is $\boldsymbol{E}$ at time $t = 0$. To obtain the solution at a given time, specify a phase factor in all results pages and in the corresponding functions.

References for Electromagnetic Theory

1. D.K. Cheng, *Field and Wave Electromagnetics*, Addison-Wesley, Reading, Massachusetts, 1989.

2. J. Jin, *The Finite Element Method in Electromagnetics*, John Wiley & Sons, New York, 1993.

3. B.D. Popovic, *Introductory Engineering Electromagnetics*, Addison-Wesley, Reading, Massachusetts, 1971.

Electromagnetic Forces

There are several ways to compute electromagnetic forces in COMSOL Multiphysics. In the most general case, the calculation of electromagnetic forces involves the computation of volume forces acting on a body, and of surface forces originating from jumps in the electromagnetic fields on the boundaries. The volume and surface forces are derived from a general stress tensor that includes electromagnetic terms.

The derivation of the expressions for the electromagnetic stress tensor utilizes thermodynamic potential (energy) principles (Ref. 1 and Ref. 3). The distribution of electromagnetic forces in a system depends on the material. Accordingly, the techniques and expressions used when calculating electromagnetic forces are different for different types of materials.

Another technique for calculating forces using the method of virtual work is described in the section Electromagnetic Energy and Virtual Work.

In this section:

- Overview of Forces in Continuum Mechanics
- Forces on an Elastic Solid Surrounded by Vacuum or Air
- Torque
- Forces in Stationary Fields
- Forces in a Moving Body
- Electromagnetic Energy and Virtual Work

Overview of Forces in Continuum Mechanics

Cauchy's equation of continuum mechanics reads

$$\rho\frac{d^2\mathbf{r}}{dt^2} = \nabla \cdot T + \mathbf{f}_{\text{ext}}$$

where ρ is the density, $\mathbf{r}$ denotes the coordinates of a material point, T is the stress tensor, and $\mathbf{f}_{\text{ext}}$ is an external volume force such as gravity ($\mathbf{f}_{\text{ext}} = \rho\mathbf{g}$). This is the equation solved in the structural mechanics physics interfaces for the special case of a linear elastic material, neglecting the electromagnetic contributions.

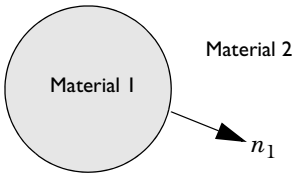
In the stationary case there is no acceleration, and the equation representing the force balance is

$$\mathbf{0} = \nabla \cdot T + \mathbf{f}_{\text{ext}}$$

The stress tensor must be continuous across a stationary boundary between two materials. This corresponds to the equation

$$\mathbf{n}_1(T_2 - T_1) = \mathbf{0}$$

where T_1 and T_2 represent the stress tensor in Materials 1 and 2, respectively, and $\mathbf{n}_1$ is the normal pointing out from the domain containing Material 1. This relation gives rise to a surface force acting on the boundary between Material 1 and 2.



In certain cases, the stress tensor T can be divided into one part that depends on the electromagnetic field quantities and one part that is the mechanical stress tensor,

$$T = T_{\text{EM}} + \sigma_{\text{M}}$$

For the special case of an elastic body, the mechanical stress tensor is proportional only to the strain and the temperature gradient. The exact nature of this split of the stress tensor into an electromagnetic and a mechanical part depends on the material model, if it can be made at all.

For more information on the mechanical stress tensor for elastic materials, see the documentation for the interfaces. For example, The Structural Mechanics Branch in the *COMSOL Multiphysics User's Guide*.

It is sometimes convenient to use a volume force instead of the stress tensor. This force is obtained from the relation

$$\mathbf{f}_{\text{em}} = \nabla \cdot T_{\text{EM}}$$

This changes the force balance equation to

$$\mathbf{0} = \nabla \cdot \sigma_M + \mathbf{f}_{em} + \mathbf{f}_{ext}$$

or, as stated in the structural mechanics physics interfaces,

$$-\nabla \cdot \sigma_M = \mathbf{f} \quad \text{where} \quad \mathbf{f} = \mathbf{f}_{em} + \mathbf{f}_{ext}$$

Forces on an Elastic Solid Surrounded by Vacuum or Air

Consider a solid (Material 1) surrounded by vacuum (Material 2). It is natural to associate the surface force on the boundary between the materials with the solid. In many applications air can be approximated by vacuum.

In practice, the equation for the force balance also needs to include an external boundary force $\mathbf{g}_{ext}$. It is nonzero on those parts of the boundary where it is necessary to compensate for the contributions to the stress tensor that you are not interested in or do not have enough information on. These contributions come from the influence of the adjacent domains. By approximating the surroundings by vacuum or air, the influence of these boundaries and their adjacent domains (that are not part of our model) on the electromagnetic fields are neglected.

On the boundary, the following equations apply:

$$\mathbf{n}_1(\tilde{T}_2 - T_1) = \mathbf{0}$$
$$\mathbf{n}_1\tilde{T}_2 = \mathbf{n}_1 T_2 + \mathbf{g}_{ext}$$

The external boundary force $\mathbf{g}_{ext}$ can represent the reaction force from another body that the solid is attached to.

The equations for the balance of forces on the solid now become

$$\nabla \cdot T_1 + \mathbf{f}_{ext} = \mathbf{0}$$
$$\mathbf{n}_1(T_2 - T_1) + \mathbf{g}_{ext} = \mathbf{0}$$

For calculating the total force $\mathbf{F}$ on the solid these equations need to be integrated over the entire solid and the solid/vacuum boundary

$$\int_{\Omega_1} (\nabla \cdot T_1 + \mathbf{f}_{ext})dV + \oint_{\partial\Omega_1} (\mathbf{n}_1(T_2 - T_1) + \mathbf{g}_{ext})dS = \mathbf{0}$$

According to Gauss' theorem:

$$\int_{\Omega_1} \nabla \cdot T_1 dV - \oint_{\partial\Omega_1} \mathbf{n}_1 T_1 dS = \mathbf{0}$$

this means that the external force

$$\mathbf{F}_{\text{ext}} = \int_{\Omega_1} \mathbf{f}_{\text{ext}} dV + \oint_{\partial\Omega_1} \mathbf{g}_{\text{ext}} dS$$

is needed to balance the term for the boundary integral of the stress tensor in the surrounding vacuum

$$\mathbf{F} = \oint_{\partial\Omega_1} \mathbf{n}_1 T_2 dS$$

to keep the solid stationary. That is $\mathbf{F}_{\text{ext}} + \mathbf{F} = \mathbf{0}$. If the external forces are suddenly removed, the solid is no longer stationary, but $\mathbf{F}$ causes the solid to begin to move with an initial acceleration according to

$$m\mathbf{a} = \int_{\Omega_1} \rho \frac{d^2\mathbf{r}}{dt^2} dV = \mathbf{F}$$

where m is the total mass and $\mathbf{a}$ is the acceleration of the solid.

To summarize, the total force, $\mathbf{F}$, is computed as a boundary integral of the stress tensor in vacuum on the outside of the solid. To obtain this result, the contribution from the air pressure gradient has been neglected. This is equivalent of assuming that $\nabla \cdot T_2 = 0$. A more detailed treatment shows that the pressure gradient contributes with a lifting (buoyancy) force on the solid.

Torque

The torque in the case of Forces on an Elastic Solid Surrounded by Vacuum or Airis given by

$$\mathbf{M}_{\text{O}} = \oint_{\partial\Omega_1} (\mathbf{r} - \mathbf{r}_{\text{O}}) \times (\mathbf{n}_1 T_2) dS$$

where $\mathbf{r}_{\text{O}}$ is a point on the axis of rotation. This follows from a derivation similar to the one made for forces.

The electromagnetic fields are stationary if

$$\frac{\partial \mathbf{B}}{\partial t} = \mathbf{0}$$

$$\frac{\partial \mathbf{D}}{\partial t} = \mathbf{0}$$

that is, if the fields vary so slowly that the contributions from induced currents and displacement currents can be neglected.

Also assume that the objects modeled are not moving $\mathbf{v} = \mathbf{0}$ so that there is no contributions from Lorentz forces. These are treated later on.

THE ELECTROMAGNETIC STRESS TENSOR

Model

To apply the stress tensor in air to calculate the total force and torque on a magnetizable rod close to a permanent magnet, see Permanent Magnet: Model Library path **ACDC_Module/Magnetostatics/permanent_magnet**.

The expressions for the stress tensor in a general electromagnetic context stems from a fusion of material theory, thermodynamics, continuum mechanics, and electromagnetic field theory. With the introduction of thermodynamic potentials for mechanical, thermal, and electromagnetic effects, explicit expressions for the stress tensor can be derived in a convenient way by forming the formal derivatives with respect to the different physical fields (Ref. 1 and Ref. 3). Alternative derivations can be made for a vacuum (Ref. 4) but these cannot easily be generalized to polarized and magnetized materials.

Air and Vacuum

For air, the stress tensor is

$$T_2 = -pI - \left(\frac{\varepsilon_0}{2}\mathbf{E}\cdot\mathbf{E} + \frac{1}{2\mu_0}\mathbf{B}\cdot\mathbf{B}\right)I + \varepsilon_0\mathbf{E}\mathbf{E}^T + \frac{1}{\mu_0}\mathbf{B}\mathbf{B}^T$$

where p is the air pressure, I is the identity 3-by-3 tensor (or matrix), and $\mathbf{E}$ and $\mathbf{B}$ are 3-by-1 vectors. In this expression of the stress tensor, air is considered to be nonpolarizable and nonmagnetizable. When air is approximated by vacuum, $p = 0$.

This expression, with $p = 0$, of the stress tensor is also known as the Maxwell stress tensor.

Using the fact that, for air, $\mathbf{D} = \varepsilon_0\mathbf{E}$ and $\mathbf{B} = \mu_0\mathbf{H}$ the expression for the stress tensor can be written as

$$T_2 = -pI - \left(\frac{1}{2}\mathbf{E}\cdot\mathbf{D} + \frac{1}{2}\mathbf{H}\cdot\mathbf{B}\right)I + \mathbf{E}\mathbf{D}^T + \mathbf{H}\mathbf{B}^T$$

The equation for the balance of forces becomes

$$\mathbf{0} = \nabla\cdot\left(-pI - \left(\frac{1}{2}\mathbf{E}\cdot\mathbf{D} + \frac{1}{2}\mathbf{H}\cdot\mathbf{B}\right)I + \mathbf{E}\mathbf{D}^T + \mathbf{H}\mathbf{B}^T\right) + \mathbf{f}_{\text{ext}}$$

Maxwell's equations in free space give that the contribution of the electromagnetic part of the stress tensor is zero, and the resulting expression is

$$\mathbf{0} = -\nabla p + \mathbf{f}_{\text{ext}}$$

Thus, using the same terminology as earlier, $\mathbf{f}_{\text{em}} = \mathbf{0}$ for air, with $\sigma_{\text{M}} = -pI$. In the derivation of the total force on an elastic solid surrounded by vacuum or air, the approximation $\nabla p = 0$ has been used.

When operating with the divergence operator on the stress tensor, the relation

$$\nabla\cdot\left(\mathbf{E}\mathbf{E}^T - \frac{1}{2}\mathbf{E}\cdot\mathbf{E}I\right) = \mathbf{E}(\nabla\cdot\mathbf{E}) - \mathbf{E}\times(\nabla\times\mathbf{E})$$

is useful (and similarly for $\mathbf{B}$). From the right-hand side it is clear (using Maxwell's equations) that this is zero for stationary fields in free space.

Consider again the case of a solid surrounded by air. To compute the total force, the projection of the stress tensor on the outside of the solid surface is needed,

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

where $\mathbf{n}_1$ is the surface normal, a 1-by-3 vector, pointing out from the solid. This expression can be used directly in the boundary integral of the stress tensor for calculating the total force $\mathbf{F}$ on the solid.


Model

For an example of how to compute the total force on two parallel wires either by integrating the volume force or by integrating the stress tensor on the surrounding surface, see Electromagnetic Forces on Parallel Current-Carrying Wires: Model Library path **ACDC_Module/Verification_Models/parallel_wires**.

A material that is nonpolarizable and nonmagnetizable ($\mathbf{P} = \mathbf{0}$ and $\mathbf{M} = \mathbf{0}$) is called a *pure conductor*. This is not necessarily equivalent to a perfect conductor, for which $\mathbf{E} = \mathbf{0}$, but merely a restriction on the dielectric and magnetic properties of the material. The stress tensor becomes identical to the one for air, except for $-pI$ being replaced by the purely mechanical stress tensor σ_M:

$$T_1 = \sigma_M - \left(\frac{1}{2}\mathbf{E}\cdot\mathbf{D} + \frac{1}{2}\mathbf{H}\cdot\mathbf{B}\right)I + \mathbf{E}\mathbf{D}^T + \mathbf{H}\mathbf{B}^T$$

where $\mathbf{D} = \varepsilon_0\mathbf{E}$ and $\mathbf{B} = \mu_0\mathbf{H}$.

The situation is slightly different from the case of air because there can be currents and volume charges in the conductor. The current density is

$$\mathbf{J} = \nabla\times\mathbf{H} = \frac{1}{\mu_0}\nabla\times\mathbf{B}$$

and the volume charge density

$$\rho = \nabla\cdot\mathbf{D} = \varepsilon_0\nabla\cdot\mathbf{E}$$

The equation for the balance of forces now becomes

$$\mathbf{0} = \nabla\cdot\sigma_M + \rho\mathbf{E} + \mathbf{J}\times\mathbf{B} + \mathbf{f}_{ext}$$

and this means that

$$\mathbf{f}_{em} = \rho\mathbf{E} + \mathbf{J}\times\mathbf{B}$$

General Elastic Material

For an elastic solid, in the general case of a material that is both dielectric and magnetic (nonzero $\mathbf{P}$ and $\mathbf{M}$), the stress tensor is given by the expression

$$T_1 = \sigma(\mathbf{E}, \mathbf{B}) - \left(\frac{\varepsilon_0}{2}\mathbf{E}\cdot\mathbf{E} + \frac{1}{2\mu_0}\mathbf{B}\cdot\mathbf{B} - \mathbf{M}\cdot\mathbf{B}\right)I$$
$$+ \varepsilon_0\mathbf{E}\mathbf{E}^T + \frac{1}{\mu_0}\mathbf{B}\mathbf{B}^T + \mathbf{E}\mathbf{P}^T - \mathbf{M}\mathbf{B}^T$$

where in $\sigma(\mathbf{E}, \mathbf{B})$ the dependence of $\mathbf{E}$ and $\mathbf{B}$ has not been separated out. Thus σ is not a purely mechanical stress tensor in this general case. Different material models give different appearances of $\sigma(\mathbf{E}, \mathbf{B})$. The electromagnetic contributions to $\sigma(\mathbf{E}, \mathbf{B})$ typically represent pyroelectric, pyromagnetic, piezoelectric, piezomagnetic, dielectric, and magnetization effects. The expression for the stress tensor in vacuum, air, and pure conductors can be derived from this general expression by setting $\mathbf{M} = \mathbf{P} = \mathbf{0}$.

T_1 must be symmetric. The terms $\mathbf{E}\mathbf{P}^T$ and $-\mathbf{M}\mathbf{B}^T$ are symmetric in the case of a linear dielectric and magnetic material because

$$\mathbf{P} = \varepsilon_0\chi_e\mathbf{E}$$
$$\mathbf{M} = \chi_B\mathbf{B}$$

Here, the magnetic susceptibility χ_B differs slightly from the classical χ_m. The other explicit terms are all symmetric, as is $\sigma(\mathbf{E}, \mathbf{B})$. In the general case this imposes constraints on the properties of $\sigma(\mathbf{E}, \mathbf{B})$. For a nonlinear material $\sigma(\mathbf{E}, \mathbf{B})$ might need to include terms such as $-\mathbf{E}\mathbf{P}^T$ or $+\mathbf{M}\mathbf{B}^T$ to compensate for asymmetric $\mathbf{E}\mathbf{P}^T$ or $-\mathbf{M}\mathbf{B}^T$.

To instantiate the stress tensor for the general elastic case, an explicit material model including the magnetization and polarization effects is needed. Such material models can easily be found for piezoelectric materials (Ref. 3).

Forces in a Moving Body

Calculating forces in moving objects is important, especially for electric motors and other moving electromagnetic devices. When performing the computations in a coordinate system that moves with the object, the electromagnetic fields are transformed. The most well-known relation for moving objects is the one for the electric field. The transformed quantity of the electric field is called the *electromotive intensity.*

FIELD TRANSFORMATIONS AND GALILEI INVARIANTS

Assume that the object modeled is moving with a constant velocity, $\mathbf{v} = \mathbf{v}_0$. The equations now take on a slightly different form that includes the Galilei invariant

versions of the electromagnetic fields. The term Galilei invariant is used due to the fact that they remain unchanged after a coordinate transformation of the type

$$\mathbf{r}' = \mathbf{r} + \mathbf{v}_0 t$$

In continuum mechanics, this transformation is commonly referred to as a Galilei transformation.

The Galilei invariant fields of interest are

$$\tilde{\mathbf{E}} = \mathbf{E} + \mathbf{v} \times \mathbf{B} \quad \text{(Electromotive intensity)}$$
$$\tilde{\mathbf{J}} = \mathbf{J} - \rho \mathbf{v} \quad \text{(Free conduction current density)}$$
$$\tilde{\mathbf{P}} = \frac{\partial \mathbf{P}}{\partial t} + \mathbf{v}(\nabla \cdot \mathbf{P}) - \nabla \times (\mathbf{v} \times \mathbf{P}) \quad \text{(Polarization flux derivative)}$$
$$\tilde{\mathbf{M}} = \mathbf{M} + \mathbf{v} \times \mathbf{P} \quad \text{(Lorentz magnetization)}$$
$$\tilde{\mathbf{H}} = \frac{\mathbf{B}}{\mu_0} - \varepsilon_0 \mathbf{v} \times \mathbf{E} - \tilde{\mathbf{M}} \quad \text{(Magnetomotive intensity)}$$

The electromotive intensity is the most important of these invariants. The Lorentz magnetization is significant only in materials for which neither the magnetization $\mathbf{M}$ nor the polarization $\mathbf{P}$ is negligible. Such materials are rare in practical applications. The same holds for the magnetization term of the magnetomotive intensity. Notice that the term $\varepsilon_0 \mathbf{v} \times \mathbf{E}$ is very small compared to $\mathbf{B}/\mu_0$ except for cases when $\mathbf{v}$ and $\mathbf{E}$ are both very large. Thus in many practical cases this term can be neglected.

Air and Vacuum

The stress tensor in the surrounding air or vacuum on the outside of a moving object is

$$T_2 = -pI - \left(\frac{1}{2}\mathbf{E} \cdot \mathbf{D} + \frac{1}{2}\mathbf{H} \cdot \mathbf{B}\right) I + \mathbf{E}\mathbf{D}^T + \mathbf{H}\mathbf{B}^T + (\mathbf{D} \times \mathbf{B})\mathbf{v}^T$$

There is an additional term in this expression compared to the stationary case.

Elastic Pure Conductor

The stress tensor in a moving elastic pure conductor is

$$T_1 = \sigma_M - \left(\frac{1}{2}\mathbf{E} \cdot \mathbf{D} + \frac{1}{2}\mathbf{H} \cdot \mathbf{B}\right) I + \mathbf{E}\mathbf{D}^T + \mathbf{H}\mathbf{B}^T + (\mathbf{D} \times \mathbf{B})\mathbf{v}^T$$

where $\mathbf{D} = \varepsilon_0 \mathbf{E}$ and $\mathbf{B} = \mu_0 \mathbf{H}$.

To get the equation for the balance of forces the divergence of this expression needs to be computed. This requires an introduction of an extra term in Cauchy's equation corresponding to an additional electromagnetic contribution to the linear momentum. Cauchy's equation with this extra term is

$$\rho \frac{d^2\mathbf{r}}{dt^2} + \mathbf{D} \times \mathbf{B} = \nabla \cdot T + \mathbf{f}_{\text{ext}}$$

The extra term is canceled out by the additional term in the stress tensor, and the final result is

$$\rho \frac{d^2\mathbf{r}}{dt^2} = \nabla \cdot \sigma_{\text{M}} + \rho \tilde{\mathbf{E}} + \tilde{\mathbf{J}} \times \mathbf{B} + \mathbf{f}_{\text{ext}}$$

For the case of no acceleration, with the explicit appearance of the transformed quantities,

$$\mathbf{0} = \nabla \cdot \sigma_{\text{M}} + \rho(\mathbf{E} + \mathbf{v} \times \mathbf{B}) + (\mathbf{J} - \rho \mathbf{v}) \times \mathbf{B} + \mathbf{f}_{\text{ext}}$$

The terms containing $\mathbf{v} \times \mathbf{B}$ cancel out, which yields the following equation:

$$\mathbf{0} = \nabla \cdot \sigma_{\text{M}} + \rho \mathbf{E} + \mathbf{J} \times \mathbf{B} + \mathbf{f}_{\text{ext}}$$

which is the same expression as for the stationary case.

General Elastic Material

The stress tensor for a moving general elastic material is

$$T_1 = \sigma(\tilde{\mathbf{E}}, \mathbf{B}) - \left(\frac{\varepsilon_0}{2}\mathbf{E} \cdot \mathbf{E} + \frac{1}{2\mu_0}\mathbf{B} \cdot \mathbf{B} - \tilde{\mathbf{M}} \cdot \mathbf{B}\right) I + \\ + \varepsilon_0 \mathbf{E}\mathbf{E}^T + \frac{1}{\mu_0}\mathbf{B}\mathbf{B}^T + \tilde{\mathbf{E}}\mathbf{P}^T - \tilde{\mathbf{M}}\mathbf{B}^T + \varepsilon_0(\mathbf{E} \times \mathbf{B})\mathbf{v}^T$$

The magnetization $\mathbf{M}$ and the polarization $\mathbf{P}$ occur explicitly in this expression.

To instantiate the stress tensor for the general elastic case a material model explicitly including the magnetization and polarization effects is needed.

Electromagnetic Energy and Virtual Work

Another technique to calculate forces is to derive the electromagnetic energy of the system and calculate the force by studying the effect of a small displacement. This is known as the ***method of virtual work*** or the ***principle of virtual displacement***.

The method of virtual work is used for the electric energy and magnetic energy separately for calculating the total electric or magnetic force as follows.

MAGNETIC FORCE AND TORQUE

The method of virtual work utilizes the fact that under *constant magnetic flux* conditions (Ref. 5), the total magnetic force on a system is computed as

$$\mathbf{F}_{\Phi} = -\nabla W_{\mathrm{m}}$$

If the system is constrained to rotate about an axis the torque is computed as

$$T_{\Phi} = -\frac{\partial W_{\mathrm{m}}}{\partial \varphi}$$

where φ is the rotational angle about the axis.

Under the condition of *constant currents*, the total force and torque are computed in the same way but with opposite signs,

$$\mathbf{F}_{I} = \nabla W_{\mathrm{m}}$$

$$T_{I} = \frac{\partial W_{\mathrm{m}}}{\partial \varphi}$$

ELECTRIC FORCE AND TORQUE

Under the condition of *constant charges*, the total electric force and torque on a system are computed as

$$\mathbf{F}_{Q} = -\nabla W_{\mathrm{e}}$$

$$T_{Q} = -\frac{\partial W_{\mathrm{e}}}{\partial \varphi}$$

Under the condition of *constant potentials*, the total electric force and torque on a system are computed as

$$\mathbf{F}_V = \nabla W_\mathrm{e}$$

$$T_V = \frac{\partial W_\mathrm{e}}{\partial \varphi}$$


Note

The method of virtual work can be employed by using the features for deformed mesh and sensitivity analysis in COMSOL Multiphysics.

See The Deformed Geometry and Moving Mesh Interfaces and Sensitivity Analysis in the *COMSOL Multiphysics User's Guide*.

Electromagnetic Quantities

The table below shows the symbol and SI unit for most of the physical quantities that appear in the AC/DC Module.

TABLE 2-1: ELECTROMAGNETIC QUANTITIES

QUANTITY	SYMBOL	SI UNIT	ABBREVIATION
Angular frequency	ω	radian/second	rad/s
Attenuation constant	α	meter^{-1}	m^{-1}
Capacitance	C	farad	F
Charge	q	coulomb	C
Charge density (surface)	ρ_s	coulomb/meter^2	C/m^2
Charge density (volume)	ρ	coulomb/meter^3	C/m^3
Current	I	ampere	A
Current density (surface)	$\mathbf{J}_s$	ampere/meter	A/m
Current density (volume)	$\mathbf{J}$	ampere/meter^2	A/m^2
Electric displacement	$\mathbf{D}$	coulomb/meter^2	C/m^2
Electric field	$\mathbf{E}$	volt/meter	V/m
Electric potential	V	volt	V
Electric susceptibility	χ_e	(dimensionless)	-
Electrical conductivity	σ	siemens/meter	S/m
Energy density	W	joule/meter^3	J/m^3
Force	$\mathbf{F}$	newton	N
Frequency	ν	hertz	Hz
Impedance	Z, η	ohm	Ω
Inductance	L	henry	H
Magnetic field	$\mathbf{H}$	ampere/meter	A/m
Magnetic flux	Φ	weber	Wb
Magnetic flux density	$\mathbf{B}$	tesla	T
Magnetic potential (scalar)	V_m	ampere	A
Magnetic potential (vector)	$\mathbf{A}$	weber/meter	Wb/m
Magnetic susceptibility	χ_m	(dimensionless)	-
Magnetization	$\mathbf{M}$	ampere/meter	A/m

TABLE 2-1: ELECTROMAGNETIC QUANTITIES

QUANTITY	SYMBOL	SI UNIT	ABBREVIATION
Permeability	μ	henry/meter	H/m
Permittivity	ε	farad/meter	F/m
Polarization	$\mathbf{P}$	coulomb/meter^2	C/m^2
Poynting vector	$\mathbf{S}$	watt/meter^2	W/m^2
Propagation constant	β	radian/meter	rad/m
Reactance	X	ohm	Ω
Relative permeability	μ_r	(dimensionless)	-
Relative permittivity	ε_r	(dimensionless)	-
Resistance	R	ohm	W
Resistive loss	Q	watt/meter^3	W/m^3
Torque	T	newton-meter	N·m
Velocity	$\mathbf{v}$	meter/second	m/s
Wavelength	λ	meter	m
Wave number	k	radian/meter	rad/m

References for the AC/DC Interfaces

1. A. Kovetz, *The Principles of Electromagnetic Theory*, Cambridge University Press, 1990.

2. Jianming Jin, *The Finite Element Method in Electromagnetics*, 2nd ed., Wiley-IEEE Press, May 2002.

3. O. Wilson, *Introduction to Theory and Design of Sonar Transducers*, Peninsula Publishing, 1988.

4. R.K. Wangsness, *Electromagnetic Fields*, 2nd ed., John Wiley & Sons, 1986.

5. D.K. Cheng, *Field and Wave Electromagnetics*, 2nd ed., Addison-Wesley, 1991.

6. O.C. Zienkiewicz, C. Emson, and P. Bettess, "A Novel Boundary Infinite Element," *International Journal for Numerical Methods in Engineering*, vol. 19, no. 3, pp. 393–404, 1983.

3

Modeling with the AC/DC Module

The goal of this chapter is to familiarize you with the modeling procedure in the AC/DC Module. Because this module is fully integrated with COMSOL Multiphysics, the modeling process is similar.

In this chapter:

Preparing for Modeling

This section is intended a guide through the selection process among the physics interfaces in the AC/DC Module and does not contain detailed interface descriptions. Several topics in the art of modeling are covered here that may not be in ordinary textbooks on electromagnetic theory. This section discusses these topics:

- What Problems Can You Solve?—Can I use the quasi-static physics interfaces or do I need wave propagation?
- Selecting the Space Dimension for the Model Geometry—Is a 2D, 3D, or axisymmetric geometry best for my model?
- Simplifying the Geometry Using Boundary Conditions—When do I need to resolve the thickness of thin shells?
- Applying Electromagnetic Sources—What sources can I use to excite the fields?
- Selecting a Study Type—Is my problem suited for time-dependent or time-harmonic (frequency domain) formulations?
- 2D Field Variables— What do you need to do to solve for a vector field in 2D?
- Meshing and Solving—What issues might arise with respect to meshing and solving?


See Also

For general guidelines for effective modeling see Overview of the Physics Interfaces and Building a COMSOL Model in the *COMSOL Multiphysics User's Guide*.

GENERAL TIPS

These general tips about modeling help you to decide what to include in a simulation and what can be done to minimize the size of a problem. Before starting to model, try to answer the following questions first:

- What is the purpose of the model?
- What information do you want to extract from the model?


Important

Models never capture all the details of reality.

Increasing the complexity of a model to make it more accurate usually makes it more expensive to simulate. A complex model is also more difficult to manage and interpret than a simple one.

It can be more accurate and efficient to use several simple models instead of a single, complex one.

What Problems Can You Solve?

The AC/DC Module interfaces handle static, time-dependent, and time-harmonic problems. The time-dependent and time-harmonic formulations use a quasi-static approximation. See Table 1-1 in Overview of the User's Guide for a list of the preset study types available by interface.

One major difference between quasi-static and high-frequency modeling is that the formulations depend on the *electrical size* of the structure. This dimensionless measure is the ratio between the largest distance between two points in the structure divided by the wavelength of the electromagnetic fields.

The quasi-static physics interfaces in this module are suitable for simulations of structures with an electrical size in the range up to 1/10. The physical assumption of these situations is that the currents and charges generating the electromagnetic fields vary so slowly in time that the electromagnetic fields are practically the same at every instant as if they had been generated by stationary sources.

When the variations in time of the sources of the electromagnetic fields are more rapid, it is necessary to solve the full Maxwell equations for high-frequency electromagnetic waves. They are appropriate for structures of electrical size 1/100 and larger. Thus, an overlapping range exists where both the quasi-static and the full Maxwell formulations can be used.

Interfaces for high-frequency electromagnetic waves are available in the RF Module.

Independent of the structure size, the AC/DC Module accommodates any case of nonlinear, inhomogeneous, or anisotropic media. It also handles materials with

properties that vary as a function of time as well as frequency-dispersive materials. Examples of applications that successfully simulate with this module include electric motors, generators, permanent magnets, induction heating devices, and dielectric heating. For a more detailed description of some of these applications, refer to the Model Library included with the module.

Selecting the Space Dimension for the Model Geometry

Most of the problems solved with COMSOL Multiphysics are three-dimensional (3D) in the real world. In many cases, it is sufficient to solve a two-dimensional (2D) problem that is close, or equivalent, to the real problem.

It is good practice to start a modeling project by building one or several 2D models before going to a 3D model.

This is because 2D models are easier to modify and solve much faster. Thus, modeling mistakes are much easier to find when working in 2D. Once the 2D model is verified, you are in a much better position to build a 3D model.

2D PROBLEMS

The following is a guide through some of the common approximations made for 2D problems. Remember that modeling in 2D usually represents some 3D geometry under the assumption that nothing changes in the third dimension.

Cartesian Coordinates

In this case you view a cross section in the *xy*-plane of the actual 3D geometry. The geometry is mathematically extended to infinity in both directions along the *z*-axis, assuming no variation along that axis. All the total flows in and out of boundaries are per unit length along the *z*-axis. A simplified way of looking at this is to assume that the geometry is extruded one unit length from the cross section along the *z*-axis. The total flow out of each boundary is then from the face created by the extruded boundary (a boundary in 2D is a line).

There are usually two approaches that lead to a 2D cross-section view of a problem:

- When it is known that there is no variation of the solution in one particular dimension
- When there is a problem where the influence of the finite extension in the third dimension can be neglected

Model

Electromagnetic Forces on Parallel Current-Carrying Wires: Model Library path **ACDC_Module/Verification_Models/parallel_wires**

The geometry has a finite width but the model neglects the (end) effects from the faces parallel to the cross section because the strongest forces are between the perpendicular faces (those seen as lines in the cross section).

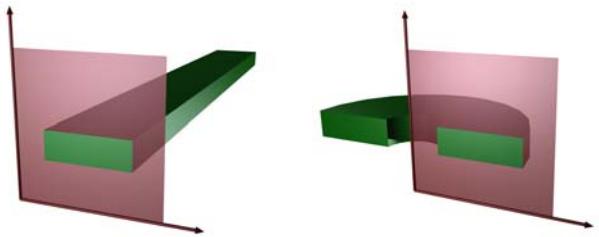


Figure 3-1: The cross sections and their real geometry for Cartesian coordinates and cylindrical coordinates (axial symmetry).

Axial Symmetry (Cylindrical Coordinates)

1D Axi

2D Axi

If the 3D geometry can be constructed by revolving a cross section about an axis, and no variations in any variable occur when going around the axis of revolution, an axisymmetric physics interface can be used.

The spatial coordinates are called r and z, where r is the radius. The flow at the boundaries is given per unit length along the third dimension. Because this dimension is a revolution, you have to multiply all flows with αr, where α is the revolution angle (for example, 2π for a full turn).


3D

This section discusses issues that should be addressed before starting to implement a 3D model.

Although COMSOL Multiphysics fully supports arbitrary 3D geometries, it is important to simplify the problem. This is because 3D problems easily get large and require more computer power, memory, and time to solve. The extra time spent on simplifying a problem is probably well spent when solving it.

Is it possible to solve the problem in 2D? Given that the necessary approximations are small, the solution is more accurate in 2D because a much denser mesh can be used. See 2D Problems if this is applicable.

Are there symmetries in the geometry and model? Many problems have planes where the solution on either side of the plane looks the same. A good way to check this is to flip the geometry around the plane, for example, by turning it upside down around the horizontal plane. You can then remove the geometry below the plane if you do not see any differences between the two cases regarding geometry, materials, and sources. Boundaries created by the cross section between the geometry and this plane need a symmetry boundary condition, which is available in all 3D physics interfaces.


Model

Eddy Currents: Model Library path **ACDC_Module/Inductive_Devices_and_Coils/eddy_currents**

Do you know the dependence in one direction so it can be replaced by an analytical function? You can use this approach either to convert 3D to 2D or to convert a layer to a boundary condition (see Simplifying the Geometry Using Boundary Conditions).

Simplifying the Geometry Using Boundary Conditions

An important technique to minimize the problem of size is to use efficient boundary conditions. Truncating the geometry without introducing large errors is one of the great challenges in modeling. Following are some ideas of how to do this in both 2D and 3D problems.

Does the solution only undergo small changes? When a model extends to infinity, it might have regions where the solution only undergoes small changes. This problem is addressed in two related steps. First, truncate the geometry in a suitable position. Second, apply a suitable boundary condition there. For static and quasi-static models, it is often possible to assume zero fields at the open boundary, provided that this is at a sufficient distance away from the sources.

Can you replace the thin layers with boundary conditions? There are several types of boundary conditions in COMSOL Multiphysics suitable for such replacements. You can, for example, replace materials with high conductivity with the shielding boundary condition, which assumes a constant potential through the thickness of the layer. If you have a magnetic material with a high relative permeability, you can also model it using the shielding boundary condition.


Model

One-Sided Magnet and Plate: Model Library path **ACDC_Module/Magnetostatics/one_sided_magnet**

Use boundary conditions for known solutions. A body with a high conductivity at high frequency has the current density confined to a thin region beneath the surface of the wire. You can often replace the current in the body by either a surface current boundary condition or an impedance boundary condition.


Model

Cold Crucible: Model Library path **ACDC_Module/Electromagnetic_Heating/cold_crucible**

Applying Electromagnetic Sources

Electromagnetic sources can be applied in many different ways. The typical options are volume sources, boundary sources, line sources, and point sources, where point sources in 2D formulations are equivalent to line sources in 3D formulations. The way sources are imposed can have an impact on what quantities can be computed from the model. For example, a point source in an electrostatics model represents a singularity, and the electric potential does not have a finite value at the position of the source.

In a COMSOL Multiphysics model, a point source has a finite but mesh-dependent potential value. Thus, it does not make sense to compute a point-to-point capacitance, because this is defined as the ratio of charge to voltage and for a point charge, the potential is not well defined. In general, using volume or boundary sources is more flexible than using line or point sources but the meshing of the source domains becomes more expensive.

Selecting a Study Type

When variations in time are present there are two main approaches to how to represent the time dependence. The most straightforward is to solve the problem in the time domain by calculating the changes in the solution for each time step. This approach can be time consuming if small time steps are necessary for the desired accuracy. It is necessary to use this approach when the inputs are transients like turn-on and turn-off sequences.

An efficient simplification is to assume that all variations in time occur as sinusoidal signals. Then the problem is time-harmonic and it can formulated as a stationary problem in the frequency domain with complex-valued solutions. The complex value represents both the amplitude and the phase of the field, while the frequency is specified as a predefined scalar input or for frequency sweeps, provided as a solver parameter. This approach is useful because, combined with Fourier analysis, it applies to all periodic signals with the exception of nonlinear problems. Examples of typical frequency domain simulations are quasi-static problems where the input variables are sinusoidal signals.

For nonlinear problems, use a frequency domain study after a *linearization* of the problem, which assumes that the distortion of the sinusoidal signal is small.

Specify a time dependent study when you think that the nonlinear influence is very strong, or if you are interested in the harmonic distortion of a sinusoidal signal. It might also be more efficient to use a time dependent study if there is a periodic input with many harmonics, like a square-shaped signal.

There are some special predefined study types for the Induction Heating multiphysics interface. This interface is based on the assumption that the magnetic cycle time is short compared to the thermal time scale (adiabatic assumption). Thus, it is associated with two predefined study types.

- Frequency-Stationary
  - Time-harmonic magnetic fields
  - Stationary heat transfer
- Frequency-Transient
  - Time-harmonic magnetic fields
  - Transient heat transfer


See Also

AC/DC Module Study Availability

2D Field Variables

When solving for a vector field in 2D, the physics interface has three options: to solve for the out-of-plane vector, the in-plane vector, or the three-component vector. Depending on the choice, the available source specification options on the domain, boundary, edge, and point levels change accordingly.

Meshing and Solving

MESH RESOLUTION

The finite element method approximates the solution within each element, using some elementary shape function that can be constant, linear, or of higher order. Depending on the element order in the model, a finer or coarser mesh is required to resolve the solution. In general, there are three problem-dependent factors that determine the necessary mesh resolution:

Is the variation in the solution due to geometrical factors? The mesh generator automatically generates a finer mesh where there is a lot of fine geometrical details. Try to remove such details if they do not influence the solution because they produce a lot of unnecessary mesh elements.

Is the skin effect or the field variation due to losses? It is easy to estimate the skin depth from the conductivity, permeability, and frequency. You need at least two linear elements per skin depth to capture the variation of the fields. If you do not study the

skin depth, you can replace regions with a small skin depth with a boundary condition, thereby saving elements.

What is the wavelength? To resolve a wave properly, it is necessary to use about 10 linear (or 5 2nd-order) elements per wavelength. Keep in mind that the wavelength might be shorter in a dielectric medium.

SELECTING A SOLVER

In most cases the solver that COMSOL Multiphysics suggests can be used. The choice of solver is optimized for the typical case for each physics interface and study type in the AC/DC Module. However, in special cases the solver settings might need fine tuning. This is especially important for 3D problems because they use a large amount of memory. For large 3D problems, a 64-bit platform may be required.


See Also

In the *COMSOL Multiphysics User's Guide*:

- Meshing
- Solvers and Study Types

Infinite Elements

In this section:


Note

For more information about this feature, see About Infinite Element Domains and Perfectly Matched Layers in the *COMSOL Multiphysics User's Guide*.

Modeling Unbounded Domains

Many environments modeled with finite elements are unbounded or open, meaning that the fields extend toward infinity. The easiest approach to modeling an unbounded domain is to extend the simulation domain "far enough" that the influence of the terminating boundary conditions at the far end becomes negligible. This approach can create unnecessary mesh elements and make the geometry difficult to mesh due to large differences between the largest and smallest object.

Another approach is to use *infinite elements*. There are many implementations of infinite elements available, and the elements used in this module are often referred to as *mapped infinite elements* (see Ref. 6). This implementation maps the model coordinates from the local, finite-sized domain to a stretched domain. The inner boundary of this stretched domain coincides with the local domain, but at the exterior boundary the coordinates are scaled toward infinity.

The principle can be explained in a one-coordinate system, where this coordinate represents Cartesian, cylindrical, or spherical coordinates. Mapping multiple coordinate directions (for Cartesian and cylindrical systems only) is just the sum of the individual coordinate mappings.

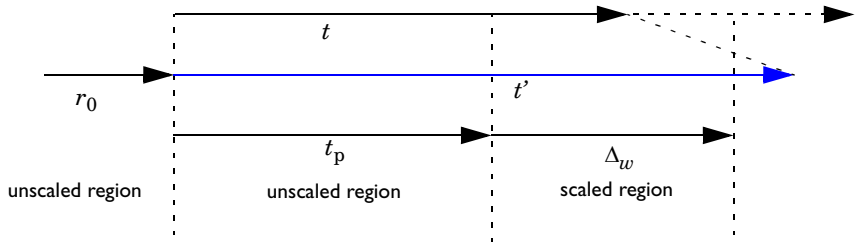


Figure 3-2: The coordinate transform used for the mapped infinite element technique. The meaning of the different variables are explained in the text.

Figure 3-2 shows a simple view of an arbitrary coordinate system. The coordinate r is the unscaled coordinate that COMSOL Multiphysics draw the geometry in (reference system). The position r_0 is the new origin from where the coordinates are scaled, t_p is the coordinate from this new origin to the beginning of the scaled region also called the *pole distance*, and Δ_w is the unscaled length of the scaled region. The scaled coordinate, t', approaches infinity when t approaches $t_p + \Delta_w$. To avoid solver issues with near infinite values, it is possible to change the infinite physical width of the scaled region to a finite large value, Δ_{pw}. The true coordinate that the PDEs are formulated in is given by

$$r' = r_0 + t'$$

where t' comes from the formula

$$t' = t_p \frac{\Delta_w}{\Delta_p - \gamma(t - t_p)}$$

$$\gamma = 1 - \frac{t_p}{\Delta_{pw} - t_p}$$

The pole distance, t_p, and the physical width of the infinite element region, Δ_{pw}, are input parameters for the region. The software automatically computes the transform for infinite element regions that are Cartesian, cylindrical, or spherical.

Important	There is no check that the geometry of the region is correct, so it is important to draw a proper geometry and select the corresponding region type.

The following figures show typical examples of infinite element regions that work nicely for each of the infinite element types. These types are:

- Stretching in Cartesian coordinate directions, labeled **Cartesian**
- Stretching in cylindrical directions, labeled **Cylindrical**
- Stretching in spherical direction, labeled **Spherical**
- User-defined coordinate transform for general infinite elements, labeled **General**

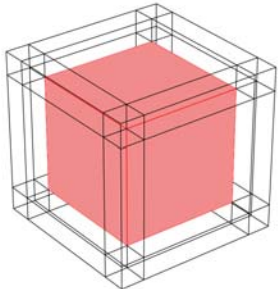


Figure 3-3: A cube surrounded by typical infinite-element regions of Cartesian type.

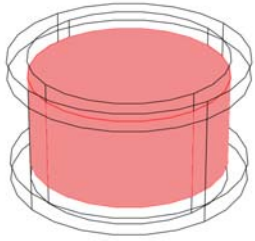


Figure 3-4: A cylinder surrounded by typical cylindrical infinite-element regions.

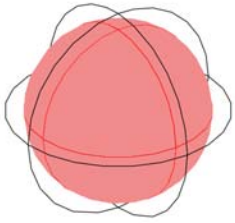


Figure 3-5: A sphere surrounded by a typical spherical infinite-element region.

If other shapes are used for the infinite element regions not similar to the shapes shown in the previous figures, it might be necessary to define the infinite element parameters manually.

The poor element quality causes poor or slow convergence for iterative solvers and make the problem ill-conditioned in general. Especially vector element formulations like the ones using two or more components of the magnetic vector potential are sensitive to low element quality. For this reason it is strongly recommended to use swept meshing in the infinite element domains. The sweep direction should be selected the same as the direction of scaling. For Cartesian infinite elements in regions with more than one direction of scaling it is recommended to first sweep the mesh in the domains with only one direction of scaling, then sweep the domains with scaling in two directions, and finish by sweeping the mesh in the domains with infinite element scaling in all three direction.

GENERAL STRETCHING

With manual control of the stretching, the geometrical parameters that defines the stretching are added as **Manual Scaling** subnodes. These subnodes have no effect unless the type of the **Infinite Elements** node is set to **General**. Each **Manual Scaling** subnode has three parameters:

- Scaling direction, which sets the direction from the interface to the outer boundary.
- Geometric width, which sets the width of the region.
- Coordinate at interface, which sets an arbitrary coordinate at the interface.

When going from any of the other types to the General type, subnodes that represent stretching of the previous type are added automatically.

Known Issues When Modeling Using Infinite Elements

Be aware of the following when modeling with infinite elements:

Use of One Single Infinite Elements Node

Use a separate Infinite Elements node for each isolated infinite element domain. That is, to use one and the same Infinite Elements node, all infinite element domains must be in contact with each other. Otherwise the infinite elements do not work properly.

Element Quality

The coordinate scaling resulting from infinite elements also yields an equivalent stretching or scaling of the mesh that effectively results in a poor element quality. (The

element quality displayed by the mesh statistics feature does not account for this effect.)

The poor element quality causes poor or slow convergence for iterative solvers and make the problem ill-conditioned in general. Especially vector element formulations like the ones using two or more components of the magnetic vector potential are sensitive to low element quality. For this reason, it is strongly recommended to use swept meshing in the infinite element domains. The sweep direction should be selected the same as the direction of scaling. For Cartesian infinite elements in regions with more than one direction of scaling it is recommended to first sweep the mesh in the domains with only one direction of scaling, then sweep the domains with scaling in two directions, and finish by sweeping the mesh in the domains with infinite element scaling in all three direction.

Complicated Expressions

The expressions resulting from the stretching get quite complicated for spherical infinite elements in 3D. This increases the time for the assembly stage in the solution process. After the assembly, the computation time and memory consumption is comparable to a problem without infinite elements. The number of iterations for iterative solvers might increase if the infinite element regions have a coarse mesh.

Erroneous Results

Infinite element regions deviating significantly from the typical configurations shown in the beginning of this section can cause the automatic calculation of the infinite element parameter to give erroneous result. Enter the parameter values manually if this is the case. See General Stretching.

Use the Same Material Parameters or Boundary Conditions

The infinite element region is designed to model uniform regions extended toward infinity. Avoid using objects with different material parameters or boundary conditions that influence the solution inside an infinite element region.

Force and Torque Computations

In this section:

Calculating Electromagnetic Forces and Torques

Two methods are available to calculate electromagnetic forces and torques:

- The most general method is to use the Maxwell stress tensor.
- Another method that works for the special case of computation of magnetic forces on nonmagnetic, current-carrying domains uses a predefined physics interface variable for the *Lorentz force* distribution in a magnetic flux density **B**.

MAXWELL STRESS TENSOR

Force and torque calculations using Maxwell's stress tensor are available in the electrostatics, electric currents, magnetic fields, and magnetic and electric fields interfaces. In electrostatics and electric currents, the force is calculated by integrating

$$\mathbf{n}_1 T_2 = -\frac{1}{2}\mathbf{n}_1(\mathbf{E}\cdot\mathbf{D}) + (\mathbf{n}_1\cdot\mathbf{E})\mathbf{D}^T \tag{3-1}$$

on the surface of the object that the force acts on.

In the magnetic fields interface, the expression

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

is integrated on the surface to obtain the force. In the magnetic and electric fields interface, both expressions are included. **E** is the electric field, **D** the electric displacement, **H** the magnetic field, **B** the magnetic flux density, and $\mathbf{n}_1$ the outward normal from the object. For a theoretical discussion about the stress tensor see Electromagnetic Forces.

LORENTZ FORCES

The *Lorentz force* is defined as $\mathbf{F} = \mathbf{J} \times \mathbf{B}$. The Lorentz force is very accurate for electromagnetic force calculations in electrically conducting domains. The Lorentz

force variables are available both in domains and on boundaries (in the case of surface currents).

Model Examples—Electromagnetic Forces

There are a number of examples in the AC/DC Module Model Library showing how to calculate electromagnetic forces in different situations.

The Electromagnetic Forces on Parallel Current-Carrying Wires model uses both Maxwell's stress tensor and the Lorentz force method to compute magnetic forces. It shows how to compute the total force on a device by integrating the volume force $\mathbf{J} \times \mathbf{B}$—the most important method for calculating forces in current-carrying devices. For materials that can be described as pure conductors (see later on in this section) this method gives the exact distribution of forces inside a device. The quantity $\mathbf{J} \times \mathbf{B}$ is the Lorentz force and is available as a predefined variable on domains and boundaries. The model also illustrates how to compute the force by integrating the Maxwell stress tensor on boundaries.

The Permanent Magnet model demonstrates how to compute the total force on a magnetizable rod close to a permanent magnet by integrating the Maxwell stress tensor in the air on the outside of the rod. This is the most important method for accurately calculating the total force on magnetic devices for which the exact distribution of volume forces is not known. To retrieve the exact distribution of volume forces requires a material that describes the interactions of the magnetizations and strains. Such materials are not always available. Therefore you are often limited to compute the total force by integrating the stress tensor or using the method of virtual work.

 *Note*	None of these methods can be used to compute and visualize the force distribution inside a domain, only to compute the total force and torque in situations where the device is surrounded by air (or when this is a good approximation).

 *Model*	• Electromagnetic Forces on Parallel Current-Carrying Wires: Model Library path **ACDC_Module/Verification_Models/parallel_wires** • Permanent Magnet: Model Library path **ACDC_Module/Magnetostatics/permanent_magnet**

Coil Domains

In this section:


See Also

- The Magnetic Fields Interface
- Magnetic Fields Interface Advanced Features

About the Single-Turn Coil Domain, Multi-Turn Coil Domain, and Coil Group Domain Features

The coil domains are features that are used to simplify the set up of magnetostatics and low-frequency electromagnetic models. In many such applications, the magnetic field is generated by electric currents flowing in conductive domains (for example, cables, wires, coils, or solenoids). The coil domains can be used to easily model these structures and to translate lumped quantities (currents and voltages) into distributed quantities (current densities and electric fields).

There are three types of coil domain features, which differ by the physical system represented, the modeling details, and the applicability to a model.


2D


2D Axi

In 2D and 2D axisymmetric, the direction of the current flow in the coil is assumed to be in the out-of-plane direction. The coil domains model straight coils in 2D and circular coils in 2D axisymmetry.

The length of the coil is equal to the out-of-plane thickness in 2D and $2\pi r$ for 2D axisymmetric models.


3D

For 3D model geometries, the current flow is not easily determined. The coil domains have settings and subfeatures to solve this problem.

SINGLE-TURN COIL DOMAIN

The Single-Turn Coil Domain feature models a single, solid domain of a conducting material (for example, metal) in which the current flows. The lumped voltage and current of the coil correspond respectively to the integral of the electric field along the coil length and to the integral of the current density on a cross section. Use this feature to model a single wire with a non-negligible cross section.


Tip

Selecting unconnected domains with a Single-Turn Coil Domain connects them in parallel.


Model

Induction Currents from Circular Coils: Model Library path **ACDC_Module/Inductive_Devices_and_Coils/coil_above_plate**

MULTI-TURN COIL DOMAIN

The Multi-Turn Coil Domain feature implements a homogenized model of a coil consisting of numerous tightly-wound conducting wires, separated by an electrical insulator. The computation of the voltage and current of the coil is performed in a similar way as for the Single-Turn Coil Domain, but it also takes into account parameters such as the number of wires and the cross section area. Use this feature to model a coil containing a large number of wires without the need to model each wire individually.


Model

- Eddy Currents in a Cylinder: Model Library path **ACDC_Module/Inductive_Devices_and_Coils/coil_eddy_currents**
- Inductor in an Amplifier Circuit: Model Library path **ACDC_Module/Inductive_Devices_and_Coils/inductor_in_circuit**

COIL GROUP DOMAIN

The Coil Group Domain feature can be used in 2D and 2D axisymmetry to easily model one or more Single-Turn Coil Domain features connected in a series. Each domain in the coil domain selection is considered a separate turn of the coil.


Note

When a Coil Group Domain selection only consists of a single domain, then it is effectively the same as a Single-Turn Coil Domain.

The Coil Group Domain automatically sets up the relations between the variables associated to each domain. Current and voltage variables for each domain (or turn) are computed the same as for the Single-Turn Coil Domain, while the total variables for the coil are computed using the rules for series-connection—the total voltage is the sum of the voltages, while the total current is equal to the current flowing in each domain.

Use this feature in 2D and 2D axisymmetric models for coils with non-negligible cross sections that cross the modeling plane more than once.

About the Coil Name

For all types of coil features, a Coil Name is entered to allow the identification of the feature and the variables it creates. By default, this is set to a unique number, but can be changed as long as it is unique between all the coil features in a model.


See Also

- Multi-Turn Coil Domain
- Single-Turn Coil Domain
- Coil Group Domain

Coil Excitation

The current density flowing in the coil domain is computed from a lumped quantity that constitutes the coil excitation. The choice of this quantity can be done by setting the coil excitation parameter, while the value is specified in the associated text box. All the coil features can be excited either with a *current* or a *voltage*. The supplied value or expression is translated to a current density or electric field applied to the domain according to the coil model used. Coil features can also retrieve the value of the current

or the voltage from a circuit connection, if the parameter coil excitation is set to *Circuit (current)* or *Circuit (voltage)*, respectively.


Tip

Circuit (current) excitation works similarly to the **Current** excitation, but in this case the inputs are provided by a circuit connection.

Circuit (voltage) excitation works similarly to the **Voltage** excitation, but in this case the inputs are provided by a circuit connection.

CURRENT EXCITATION

Single-Turn Coil Domains

When specifying a total current I_{coil}, the out-of-plane component of the current density is defined as:

$$J^e = \frac{\sigma V}{L} \tag{3-2}$$

where L is equal to the physics interface thickness d for 2D models and $2\pi r$ for 2D axially symmetric models, and V is an unknown applied potential. The potential V is solved for using an additional algebraic equation, which constrains the total integrated current to be equal to the current value I_{coil} specified.

Multi-Turn Coil Domains

When specifying a total current I_{coil}, the component of the current density is defined in the direction of the wires as in Equation 3-3 where N is the number of turns which are specified and A is the total cross section area of the coil domain.

$$J_{\text{e}} = \frac{NI_{\text{coil}}}{A} \tag{3-3}$$

Coil Group Domains

When specifying a total coil current I_{coil}, the out-of-plane component of the current density is defined as:

$$J_{\text{e}} = \frac{\sigma V_i}{L}$$

where L is equal to the physics interface thickness d for 2D models and equal to $2\pi r$ for 2D axially symmetric models, and V_i is an unknown applied potential on the i^{th} turn of the coil. The potential V_i is solved for using an additional algebraic equation,

which constrains the total integrated current to be equal to the current value I_{coil} that is specified:

$$\int J dS = I_{\text{coil}}$$

VOLTAGE EXCITATION

Single-Turn Coil Domains

When specifying a total voltage V_{coil}, the out-of-plane component of the current density is defined as:

$$J^e = \frac{\sigma V_{\text{coil}}}{L} \tag{3-4}$$

where V_{coil} is the applied voltage specified, and L is equal to the physics interface's thickness d for 2D models and equal to $2\pi r$ for 2D axially symmetric models.

Multi-Turn Coil Domains

When specifying a total voltage V_{coil}, the component of the current density in the direction of the wires is defined as in Equation 3-5 and Equation 3-6.

$$J_{\text{e}} = \frac{N(V_{\text{coil}} + V_{\text{ind}})}{AR_{\text{coil}}} \tag{3-5}$$

where V_{coil} is the applied voltage which is specified, A is the total cross-sectional area of the coil domain, N is the number of turns specified, R_{coil} is the total resistance of the coil calculated as

$$\int_A \frac{NL}{\sigma_{\text{coil}} a_{\text{coil}} A} \tag{3-6}$$

where L is equal to the physics interface's thickness d for 2D models and equal to $2\pi r$ for 2D axially symmetric models. The expression $\sigma_{\text{coil}} a_{\text{coil}}$ is the product of the wire bulk conductivity and wire cross-section area. V_{ind} is the induced voltage calculated by integrating the electric field along the coil.

Coil Group Domains

When specifying a total voltage V_{coil}, the out-of-plane component of the current density is defined as:

$$J_e = \frac{\sigma V_i}{L}$$

where V_i is an unknown applied potential on the i^{th} turn of the coil, and L is equal to the physics interface thickness d for 2D models and equal to $2\pi r$ for 2D axially symmetric models. The applied potentials are computed through the integral constraint:

$$\int J_e dS = I_{coil}$$

Unlike the fixed current option, the coil current I_{coil} is unknown. The coil current is computed using the constraint:

$$V_{coil} = \sum_{i=1}^{N} V_i$$

where V_{coil} is the user-defined voltage drop across the coil, V_i is the individual applied potentials, and N is the number of turns in the coil group.

POWER EXCITATION

In 2D and 2D axisymmetric models, Power is an option available for the coil excitation parameter. When **Power** is selected, the feature sets up the equation as specified in the Current Excitation section, plus the constraint:

$$\frac{1}{2}\text{realdot}(V_{coil}, I_{coil}) = P_{coil}$$

between the coil current and voltage. Select this option to specify the input power for the coils. The problem becomes nonlinear with these settings.


Note

The **Power** option is not available for 3D models due to the increased complexity of the problem compared to 2D models.


Important

This nonlinear system of equations requires special solver settings in order to converge. COMSOL automatically adds these solver settings when Power is selected.

Be aware that, in general, the values of the voltage and the current may not uniquely be determined by this constraint. Particularly in the frequency domain, the absolute phase of the quantities can be arbitrary. This indeterminacy can have an impact on the solution process. See Compile Equations in the *COMSOL Multiphysics Reference Guide* for a possible solution to this problem.


See Also

- Multi-Turn Coil Domain
- Single-Turn Coil Domain
- Coil Group Domain
- Solver Features in the *COMSOL Multiphysics Reference Guide*

In the *COMSOL Multiphysics User's Guide:*

- Solvers and Study Types
- The Realdot Operator

Lumped Parameter Calculations

All the types of coil domains define lumped variables during the solution. These variables are defined in the global scope and have the template `<variable>coil_<name>`, where `<variable>` is the basic variable name (V for voltage, Z for impedance, and so forth) and `<name>` is the string specified in the **Coil name** field. These variables can be evaluated during the analysis phase or used in expressions; in this case, the physics interface identifier (for example, `mf` for the Magnetic Fields interface) must be added to the beginning of the variable name. A brief description of each of the variables follows.

THE VOLTAGE, CURRENT, AND POWER VARIABLES

The coil features define the variables V_{coil}, I_{coil} and P_{coil}. The variable corresponding to the quantity used for the coil excitation contains the input value, while the others have values computed from the solution according to the coil model.


Note

For the Coil Group Domain feature, V_{coil} is the total voltage across the coil, that is, the sum of the voltages across each domain.

STATIONARY AND TIME DEPENDENT STUDIES

In stationary and time dependent studies, the coil feature defines the coil resistance variable, R_{coil}, with the formula:

$$R_{coil} = \frac{V_{coil}}{I_{coil}}$$

FREQUENCY DOMAIN STUDIES

In frequency domain studies, the following lumped variables are defined:

Impedance

$$Z_{coil} = \frac{V_{coil}}{I_{coil}}$$

Resistance

$$R_{coil} = \text{real}(Z_{coil})$$

Inductance

$$L_{coil} = \frac{\text{imag}(Z_{coil})}{\omega}$$

Reactance

$$X_{coil} = \text{imag}(Z_{coil})$$

Admittance

$$Y_{coil} = Z_{coil}^{-1}$$

FREQUENCY DOMAIN, PERTURBATION STUDIES

In frequency domain perturbation studies (small-signal analysis), the parameters defined are the same as in the frequency domain study, but the impedance is computed using the harmonic voltage and current around the linearization point, that is:

$$Z_{\text{coil}} = \frac{\text{lindev}(V_{\text{coil}})}{\text{lindev}(I_{\text{coil}})}$$

- The Lindev Operator in the *COMSOL Multiphysics User's Guide*
- Multi-Turn Coil Domain
- Single-Turn Coil Domain
- Coil Group Domain

Using Coils in 3D Models

The coil features require additional settings in 3D models to determine the geometry and the direction of the current flow.

SINGLE-TURN COIL DOMAIN

A 3D Single-Turn Coil Domain represents a solid conducting domain, typically a wire or a coil, with a non-negligible cross section. To enforce the current conservation in the domain, an additional dependent variable with the dimension of an electric potential (SI unit: V) is added to the problem and the continuity equation for the current is introduced in the system of equations. This variable is referred to as the *coil potential*, but it is only loosely related to the electrostatic potential and it should be considered a help variable rather than representing a tangible physical quantity. In the Single-Turn Coil Domain node, it is possible to specify the material properties that are used in the continuity equation.

The excitation is applied by means of specialized subfeatures: a Boundary Feed subfeature applies constraints on the coil potential to an external boundary, while a Ground subfeature enforces the coil potential to be zero on the selected boundaries.

Important	To ensure that the current continuity equation has a physical solution, terminate a coil domain on external boundaries only.

A Gap Feed subfeature models a thin gap in the conductive domain across which a difference of potential or a current is applied. This feature should be applied on internal boundaries to the conductive domain and is useful for modeling closed loops.

Important	For the Single-Turn Coil Domain feature, only one active feed feature can be used at a time. That is, either a Boundary Feed or Gap Feed feature can be used, not both.

Important	The Gap Feed should be used with care in high frequency modeling as there will be a mesh dependent displacement current density flowing in the elements just outside the bounding edges of the Gap Feed boundary. If these elements have non zero conductivity, this problem may appear even at lower frequencies.

MULTI-TURN COIL DOMAIN

A Multi-Turn Coil Domain used in 3D models requires additional settings compared to 2D models. A Coil Type parameter is used to specify the geometry of the coil (that is, the direction of the wires and the length of the coil). Select a **Linear**, **Circular**, **Numeric**, or **User defined** coil.

Linear Coil Types

In a *linear coil*, the wires are straight and parallel. To specify the direction of the wires, right-click the **Multi-Turn Coil Domain** node, add a Reference Edge subfeature and select a straight edge or a group of collinear straight edges along the entire length of the coil. The coil direction is taken to be the tangential vector to the edges, while the coil length is the total length of the edges.

Circular Coil Types

In a *circular coil*, the wires are wound in circles around a common axis. To specify the direction of the wires, right-click the **Multi-Turn Coil Domain** node, add a Reference Edge subfeature and select a group of edges forming a circle with the same axis as the coil. The wires are assumed to be wound around the circle's axis, while the length of the coil is taken from the length of the edges. To obtain the best approximation possible, ensure that the circle radius is close to the average radius of the circular coil.

Numerical Coil Types

In a *numerical coil*, the current path is computed numerically in an additional study step during the solution. This allows the modeling of coils having complex shapes. To set up the numerical computation of the current flow in a coil, additional steps are required. Right-click the **Multi-Turn Coil Domain** node and add an Automatic Current Calculation subfeature. This subfeature defines the dependent variable and the equations needed for the computation, and takes care of the coupling with the parent Multi-Turn Coil Domain feature.

To define boundary conditions for the current computation, right-click the **Automatic Current Calculation** node and add the Electric Insulation, Input, and Output subfeatures. The Electric Insulation feature should be applied on the boundaries delimiting the coil domain, and it constrains the coil wires to be parallel to the boundary. The Input and Output features work similarly, constraining the wires to be orthogonal to the boundary, and together define the direction of the current flow (from Input to Output). A single Input feature can also applied to an internal boundary if the coil domain is a closed loop (that is, if Input and Output should be on the same boundary). For the problem to be correctly set-up, exactly one Input feature must be present.

To complete the set up, add a **Coil Current Calculation** study step to the study, before the main study step.

 *Important*	Ensure that the Coil Name in the Coil Current Calculation study step is the same as the name of the numerical coil. During the solution process, the current flow direction is computed in the Coil Current Calculation step and subsequently used in the following study steps.


See Also

- Computing Coil Currents
- Coil Current Calculation Study in the *COMSOL Multiphysics User's Guide*

Computing Coil Currents

In 3D models, it is possible to solve an eigenvalue problem for the current flow in a Multi-Turn Coil Domain that gives the current density likely produced by a bundle of conductive wires. The best results are obtained when the coil has a constant cross section, without sharp bends and bottlenecks. The eigenvalue problem is loosely based on the equation for the incompressible fluid flow, with some modifications, and is solved by the specialized **Coil Current Calculation** study step. If $\mathbf{e}$ is the vector field representing the direction of the wires, the equation solved is

$$-\nabla \cdot (s\nabla \mathbf{e}) = \lambda \mathbf{e}$$

where s is a square (3 by 3) matrix with 1 on the diagonal and a scaling value (default 0.1) in the other elements. This off-diagonal scaling value can be changed in the Automatic Current Calculation feature, but the default value should give good results in most cases. The eigenvalue solver solves for the eigenfunction with eigenvalue closest to zero. The current profile for this eigenfunction depends on the shape of the domain and the boundary conditions applied. These boundary conditions are of two kinds: Electric Insulation, that forces the normal component of the current flow to the boundary to be zero ($\mathbf{n} \cdot \mathbf{e} = 0$), and Input or Output, that set the tangential component to zero ($\mathbf{n} \times \mathbf{e} = 0$). The Input and Output features have the same effect on the equation system, but the Input feature also is used to determine the boundary on which to integrate to compute the total current. For this reason, only one Input boundary is allowed for each Automatic Current Calculation node. For a correct setup, one Input and at least one Output feature are required for the coil, if applied on external boundaries, or, as an alternative, a single Input feature can be applied to an internal boundary to create a closed loop coil. In any case, an Electric Insulation feature must be applied on all the external boundaries of the coil domain.

Once the vector field $\mathbf{e}$ is obtained from the eigenvalue problem, the coil current density vector field is computed by normalizing $\mathbf{e}$ and, if needed, changing its sign so that the current flow is inward at the Input feature. It is advised to always plot the coil direction variable after solving the eigenvalue problem to verify that the current flow has the intended profile.

Note	The **Coil Current Calculation** study step must precede the main study step (for example, a Stationary study step) in which the Multi-Turn Coil Domain is used.

Important	The **Coil Current Calculation** study step solves for the current flow in the coil feature whose Coil Name is the same as the name specified in the study step. If more Numeric coils are present in the interface, an equivalent number of study steps must be solved, and each of them must precede the main study step.

See Also	• About the Single-Turn Coil Domain, Multi-Turn Coil Domain, and Coil Group Domain Features • Solver Features in the *COMSOL Multiphysics Reference Guide*

Lumped Parameters

Lumped parameters are matrices describing electromagnetic properties such as resistance, capacitance, and inductance. In the time-harmonic case the lumped parameter matrix is either an impedance matrix or an admittance matrix depending on how the model is excited (current or voltage). In a static calculation only the resistive, capacitive, or inductive part of the lumped parameter matrix is obtained.

In this section:

Calculating Lumped Parameters with Ohm's Law

To calculate the lumped parameters, there must be at least two electrodes in the system, one of which must be grounded. Either a voltage or a current can be forced on the electrodes. After the simulation, extract the other property or the energy and use it when calculating the lumped parameter.

There are several available techniques to extract the lumped parameters. Which one to use depends on the physics interface, the parameter of interest, and how the model is solved. The overview of the techniques in this section use a 4-by-4 matrix example for the lumped parameter matrix. This represents a system of at least five electrodes, where four are used as terminals and the rest are grounded, as illustrated in Figure 3-6.

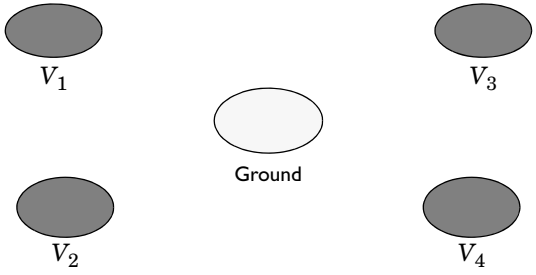


Figure 3-6: A five-electrode system with 4 terminals and one ground electrode.

If a system specifies that all electrodes are terminals, the results are redundant matrix elements. This is better understood by considering a two-electrode system. If both electrodes are declared as terminals, a 2-by-2 matrix is obtained for the system. This is clearly too many elements because there is only one unique lumped parameter between the terminals. If in addition one or more ground electrodes are declared, the system has three unique electrodes and the lumped parameter matrix becomes a 2-by-2 matrix.

FORCED VOLTAGE

If voltages are applied to the terminals, the extracted currents represent elements in the admittance matrix, **Y**. This matrix determines the relation between the applied voltages and the corresponding currents with the formula

$$\begin{bmatrix} I_1 \\ I_2 \\ I_4 \\ I_4 \end{bmatrix} = \begin{bmatrix} Y_{11} & Y_{12} & Y_{13} & Y_{14} \\ Y_{21} & Y_{22} & Y_{23} & Y_{24} \\ Y_{31} & Y_{32} & Y_{33} & Y_{34} \\ Y_{41} & Y_{42} & Y_{43} & Y_{44} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \end{bmatrix}$$

so when V_1 is nonzero and all other voltages are zero, the vector I is proportional to the first column of **Y**.

In electrostatics the current is replaced with charge and the admittance matrix is replaced with the capacitance matrix

$$\begin{bmatrix} Q_1 \\ Q_2 \\ Q_4 \\ Q_4 \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} \\ C_{21} & C_{22} & C_{23} & C_{24} \\ C_{31} & C_{32} & C_{33} & C_{34} \\ C_{41} & C_{42} & C_{43} & C_{44} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \end{bmatrix}$$

FIXED CURRENT

It might be necessary to calculate the **Z**-matrix in a more direct way. Similar to the **Y** calculation, the **Z** calculation can be done by forcing the current through one terminal at the time to a nonzero value while the others are set to zero. Then, the columns of the impedance matrix are proportional to the voltage values on all terminals:

$$\begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} & Z_{13} & Z_{14} \\ Z_{21} & Z_{22} & Z_{23} & Z_{24} \\ Z_{31} & Z_{32} & Z_{33} & Z_{34} \\ Z_{41} & Z_{42} & Z_{43} & Z_{44} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \\ I_3 \\ I_4 \end{bmatrix}$$

In magnetostatics this option means that the energy method is used; see Calculating Lumped Parameters Using the Energy Method below.

FIXED CHARGE

The Electrostatics interface can use total charge instead of total current. This gives the inverted capacitance matrix in a similar manner as the **Z** and **Y** matrices.

$$\begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} \\ C_{21} & C_{22} & C_{23} & C_{24} \\ C_{31} & C_{32} & C_{33} & C_{34} \\ C_{41} & C_{42} & C_{43} & C_{44} \end{bmatrix}^{-1} \begin{bmatrix} Q_1 \\ Q_2 \\ Q_4 \\ Q_4 \end{bmatrix}$$

Calculating Lumped Parameters Using the Energy Method

When using this method the potential or the current is nonzero on one or two terminals at a time and the energy density is extracted and integrated over the whole geometry. The following formulas show how to calculate the capacitance matrix from the integral of the electric energy density.

$$C_{ii} = \frac{2}{V_i^2}\int_\Omega W_e d\Omega \qquad V_j = \begin{cases} 0 & j \neq i \\ V_i & j = i \end{cases}$$

$$C_{ij} = \frac{1}{V_i V_j}\int_\Omega W_e d\Omega - \frac{1}{2}\left(\frac{V_i}{V_j}C_{ii} + \frac{V_j}{V_i}C_{jj}\right) \qquad V_k = \begin{cases} 0 & k \neq i, j \\ V_i & k = i \\ V_j & k = j \end{cases}$$

Calculate the inductance matrix in the same way from the magnetic energy density:

$$L_{ii} = \frac{2}{I_i^2}\int_\Omega W_m d\Omega \qquad I_j = \begin{cases} 0 & j \neq i \\ I_i & j = i \end{cases}$$

$$L_{ij} = \frac{1}{I_i I_j}\int_\Omega W_m d\Omega - \frac{1}{2}\left(\frac{I_i}{I_j}L_{ii} + \frac{I_j}{I_i}L_{jj}\right) \qquad I_k = \begin{cases} 0 & k \neq i, j \\ I_i & k = i \\ I_j & k = j \end{cases}$$

This is the technique used when **Fixed current** is selected.

Studying Lumped Parameters

To study lumped parameters, use the terminal boundary condition for each electrode. This boundary condition is available in the following interfaces and the methods described in the previous section are used to calculate the lumped parameters:

- Electrostatics. Uses a stationary study and the energy method.
- Electric Currents. Uses a stationary or frequency domain study type using the method based on Ohm's law.
- Magnetic and Electric Fields (when the electric potential is one of the dependent variables). For the stationary study the energy method is used. For the frequency domain study type, the method based on Ohm's law is used.

The lumped parameters are defined as global variables. Evaluate these from the **Derived Values** node under **Results** in the Model Builder or define 1D plot groups.

SWEEP SETTINGS AND TOUCHSTONE EXPORT

In the main node of the interface, activate a sweep to loop the excitation over the terminals in the model and calculate a lumped parameter matrix. For frequency domain models there is also an inner loop with a frequency sweep for each terminal and the lumped parameters are exported to a Touchstone file. The generated lumped parameters are in the form of an impedance or admittance matrix depending on the terminal settings. They must consistently be of either fixed voltage (for an admittance matrix) or fixed current type (for an impedance matrix).

ACCURACY

Use reaction terms to be accurate when calculating the total current over the boundary. This is necessary for the forced voltage input property. The reaction terms (representing current or charge density) come from default information stored in the solution, which gives an exact calculation of the total fluxes on boundaries with

constraints. They do not change the system of equations in any way—no special solver settings are required. The reaction terms are also stored by default. It is recommended to use forced voltage input property with reaction terms in the extraction of the lumped parameters. Lumped parameter variables based on voltage excitation are only available when reaction fluxes are included in the output. The optional current excitation performs a coupling that guarantees that the total current is equal to the specified value, although one cannot verify this without using reaction terms.

Lumped Parameter Conversion

When the impedance matrix, **Z**, or the admittance matrix, **Y**, is available it is possible to calculate all other types of lumped parameter matrices from the relations below.

$$\mathbf{S} = \mathbf{G}_{\mathrm{ref}} \cdot (\mathbf{E} - (\mathbf{Z}_{\mathrm{ref}}^{*} \cdot \mathbf{Y})) \cdot (\mathbf{E} + \mathbf{Z}_{\mathrm{ref}} \cdot \mathbf{Y})^{-1} \cdot \mathbf{G}_{\mathrm{ref}}^{-1},$$

$$\mathbf{Z} = \mathbf{Y}^{-1}, \qquad \mathbf{L} = \frac{\mathrm{Im}(\mathbf{Z})}{\omega}, \qquad \mathbf{C} = \frac{\mathrm{Im}(\mathbf{Y})}{\omega},$$

$$\mathbf{R} = \mathrm{Re}(\mathbf{Z}), \qquad \mathbf{G} = \mathrm{Re}(\mathbf{Y})$$

where **L** is the inductance, **C** is the capacitance, **R** is the resistance, and **G** is the conductance. **S** is the S-parameter. The relations also include the following matrices

$$\mathbf{E} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

$$\mathbf{Z}_{\mathrm{ref}} = \mathbf{E} \cdot Z_0$$

$$\mathbf{G}_{\mathrm{ref}} = \mathbf{E} \cdot \frac{1}{2\sqrt{|\mathrm{Re}(Z_0)|}}$$

where Z_0 is the characteristic impedance.

You can compute conversions between the impedance matrix, **Z**, the admittance matrix, **Y**, and the S-parameter matrix S in a results table using the settings in the **Global Matrix Evaluation** node, which you can add under **Results>Derived Values**. See Global Matrix Evaluation in the *COMSOL Multiphysics User's Guide* for more information.

Lumped Ports with Voltage Input

In this section:

About Lumped Ports

The ports described in the S-Parameters and Ports section require a detailed specification of the mode, including the propagation constant and field profile. In situations when the mode is difficult to calculate or when there is an applied voltage to the port, a *lumped port* might be a better choice. This is also the appropriate choice when connecting a model to an electrical circuit. For example, attach a lumped port as an internal port directly to a printed circuit board or to the transmission line feed of a device. The lumped port must be applied between two metallic objects separated by a distance much smaller than the wavelength, that is a local quasi-static approximation must be justified. This is because the concept of port or gap voltage breaks down unless the gap is much smaller than the local wavelength.

A lumped port specified as an input port calculates the impedance, Z_{port}, and S_{11} S-parameter for that port. The parameters are directly given by the relations

$$Z_{\text{port}} = \frac{V_{\text{port}}}{I_{\text{port}}}$$

$$S_{11} = \frac{V_{\text{port}} - V_{\text{in}}}{V_{\text{in}}}$$

where V_{port} is the extracted voltage for the port given by the electric field line integral between the terminals averaged over the entire port. The current I_{port} is the averaged total current over all cross sections parallel to the terminals. Ports not specified as input ports only return the extracted voltage and current.

See Also

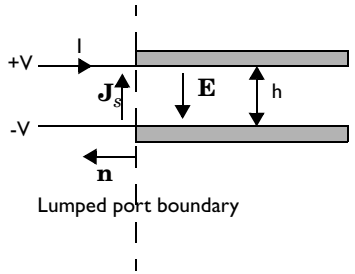
Lumped Port Parameters

Lumped Port Parameters

In transmission line theory voltages and currents are dealt with rather than electric and magnetic fields, so the lumped port provides an interface between them. The requirement on a lumped port is that the feed point must be similar to a transmission line feed, so its gap must be much less than the wavelength. It is then possible to define the electric field from the voltage as

$$V = \int_h \mathbf{E} \cdot \mathbf{dl} = \int_h (\mathbf{E} \cdot \mathbf{a}_h) dl$$

where h is a line between the terminals at the beginning of the transmission line, and the integration is going from positive (phase) V to ground. The current is positive going into the terminal at positive V.



The transmission line current can be represented with a surface current at the lumped port boundary directed opposite to the electric field.

The impedance of a transmission line is defined as

$$Z = \frac{V}{I}$$

and an analogy to this is to define an equivalent surface impedance at the lumped port boundary

$$\eta = \frac{\mathbf{E} \cdot \mathbf{a}_h}{\mathbf{J}_s \cdot (-\mathbf{a}_h)}$$

To calculate the surface current density from the current, integrate along the width, w, of the transmission line

$$I = \int_w (\mathbf{n} \times \mathbf{J}_s) \cdot \mathbf{dl} = -\int_w (\mathbf{J}_s \cdot \mathbf{a}_h) dl$$

where the integration is taken in the direction of $\mathbf{a}_h \times \mathbf{n}$. This gives the following relation between the transmission line impedance and the surface impedance

$$Z = \frac{V}{I} = \frac{\int_h (\mathbf{E} \cdot \mathbf{a}_h) dl}{-\int_w (\mathbf{J}_s \cdot \mathbf{a}_h) dl} = \eta \frac{\int_h (\mathbf{E} \cdot \mathbf{a}_h) dl}{\int_w (\mathbf{E} \cdot \mathbf{a}_h) dl} \approx \eta \frac{h}{w} \Rightarrow$$

$$\eta = Z \frac{w}{h}$$

where the last approximation assumed that the electric field is constant over the integrations. A similar relationship can be derived for coaxial cables

$$\eta = Z \frac{2\pi}{\ln \frac{b}{a}}$$

The transfer equations above are used in an impedance type boundary condition, relating surface current density to tangential electric field via the surface impedance.

$$\mathbf{n} \times (\mathbf{H}_1 - \mathbf{H}_2) + \frac{1}{\eta} \mathbf{n} \times (\mathbf{E} \times \mathbf{n}) = 2 \frac{1}{\eta} \mathbf{n} \times (\mathbf{E}_0 \times \mathbf{n})$$

where $\mathbf{E}$ is the total field and $\mathbf{E}_0$ the incident field, corresponding to the total voltage, V, and incident voltage, V_0, at the port.


Note

When using the lumped port as a circuit port, the port voltage is fed as input to the circuit and the current computed by the circuit is applied as a uniform current density—that is, as a surface current condition. Thus, an open (unconnected) circuit port is just a continuity condition.

S-Parameters and Ports

In this section:

S-Parameters in Terms of Electric Field

Scattering parameters (or S-parameters) are complex-valued, frequency dependent matrices describing the transmission and reflection of electromagnetic waves at different ports of devices like filters, antennas, waveguide transitions, and transmission lines. S-parameters originate from transmission-line theory and are defined in terms of transmitted and reflected voltage waves. All ports are assumed to be connected to matched loads, that is, there is no reflection directly at a port.

For a device with n ports, the S-parameters are

$$S = \begin{bmatrix} S_{11} & S_{12} & . & . & S_{1n} \\ S_{21} & S_{22} & . & . & . \\ . & . & . & . & . \\ . & . & . & . & . \\ S_{n1} & . & . & . & S_{nn} \end{bmatrix}$$

where S_{11} is the voltage reflection coefficient at port 1, S_{21} is the voltage transmission coefficient from port 1 to port 2, and so on. The time average power reflection/transmission coefficients are obtained as $|S_{ij}|^2$.

Now, for high-frequency problems, voltage is not a well-defined entity, and it is necessary to define the scattering parameters in terms of the electric field. For details on how COMSOL Multiphysics calculates the S-parameters, see S-Parameter Calculations.

S-Parameter Calculations in COMSOL Multiphysics: Lumped Ports

The AC/DC interfaces have a built-in support for S-parameter calculations. To set up an S-parameter study use a *Lumped Port* boundary feature for each port in the model. The lumped ports should only be used when the port width is much smaller than the wavelength.


See Also

- For more details about lumped ports, see Lumped Ports with Voltage Input.
- See Lumped Port for instructions to set up a model.

S-Parameter Variables

The AC/DC Module automatically generates variables for the S-parameters. The port names (use numbers for sweeps to work correctly) determine the variable names. If, for example, there are two lumped ports with the numbers 1 and 2 and Lumped Port 1 is the inport, the software generates the variables `S11` and `S21`. `S11` is the S-parameter for the reflected wave and `S21` is the S-parameter for the transmitted wave. For convenience, two variables for the S-parameters on a dB scale, `S11dB` and `S21dB`, are also defined using the following relation:

$$S_{11\mathrm{dB}} = 20\log 10(|S_{11}|)$$

The model and physics interface names also appear in front of the variable names so they may vary. The S-parameter variables are added to the predefined quantities in appropriate plot lists.

Connecting to Electrical Circuits

In this section:

 Model	• Inductor in an Amplifier Circuit: Model Library path **ACDC_Module/Inductive_Devices_and_Coils/inductor_in_circuit** • Tutorial Example: Modeling a 3D Inductor: Model Library path **ACDC_Module/Inductive_Devices_and_Coils/inductor_3d**

About Connecting Electrical Circuits to Physics Interfaces

This section describes the various ways electrical circuits can be connected to other physics interfaces in COMSOL Multiphysics. If you are not familiar with circuit modeling, it is recommended that you review the Theory for the Electrical Circuit Interface.

In general electrical circuits connect to other physics interfaces via one or more of three special circuit features:

- External I vs. U
- External U vs. I
- External I-Terminal

These features either accept a voltage measurement from the connecting non-circuit physics interface and return a current from the circuit interface or the other way around.


Note

The "External" features are considered "ideal" current or voltage sources by the Electrical Circuit interface. Hence, you cannot connect them directly in parallel (voltage sources) or in series (current sources) with other ideal sources. This results in the error message *The DAE is structurally inconsistent*. A workaround is to provide a suitable parallel or series resistor which can be tuned to minimize its influence on the results.

Connecting Electrical Circuits Using Predefined Couplings

In addition to these circuit features, physics interfaces in the AC/DC Module, RF Module, MEMS Module, and Plasma Module (the modules that include the Electrical Circuit interface) also contain features that provide couplings to the Electrical Circuit interface by accepting a voltage or a current from one of the specific circuit features (External I vs. U, External U vs. I, and External I-Terminal).

This coupling is typically activated when:

- A choice is made in the settings window for the non-circuit physics interface feature, which then announces (that is, includes) the coupling to the Electrical Circuit interface. Its voltage or current is then included to make it visible to the connecting circuit feature.
- A voltage or current that has been announced (that is, included) is selected in a feature node's settings window.

These circuit connections are supported in Terminal, Coils, and Lumped Ports.

Connecting Electrical Circuits by User-Defined Couplings

A more general way to connect a physics interface to the Electrical Circuit interface is to:

- Apply the voltage or current from the connecting "External" circuit feature as an excitation in the non-circuit physics interface.

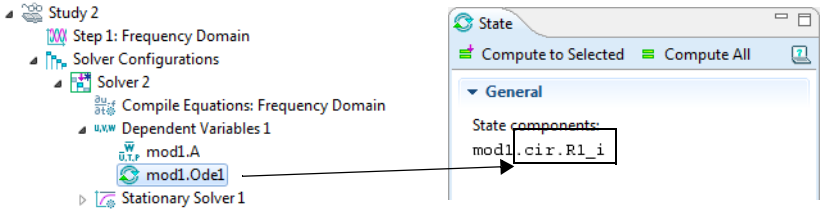
- Define your own voltage or current measurement in the non-circuit physics interface using variables, coupling operators and so forth.
- In the settings window for the Electrical Circuit interface feature, selecting the User-defined option and entering the name of the variable or expression using coupling operators defined in the previous step.

DETERMINING A CURRENT OR VOLTAGE VARIABLE NAME

To determine a current or voltage variable name, it may be necessary to look at the **Dependent Variables** node under the **Study** node. To do this:

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

2 Expand the **Solver>Dependent Variables** node and click the state node, in this example, **mod1Ode1**. The variable name is shown on the **State** settings window.




Tip

Typically, voltage variables are named `cir.Xn_v` and current variables `cir.Xn_i`, where "n" is the "External" device number, that is, `1, 2,` and so on.

Importing ECAD Files

In this section:

Overview of the ECAD Import

This section explains how to import ECAD files into COMSOL Multiphysics. An ECAD file can, for example, be a 2D layout of a printed circuit board (PCB) that is imported and converted to a 3D geometry.

EXTRUDING LAYERS

A PCB layout file holds information about all traces in several 2D drawings or layers. During import, each 2D layer is extruded to a 3D object so that all traces get a valid thickness. A standard extrude operation requires that the source plane is identical to the destination plane. This makes it impossible to extrude an entire PCB with several layers, where the source and destination planes in almost all cases do not match. It is possible to do several extrude operations, one for each layer. For complex PCBs it is not easy to put these layers together, and it might take a very long time to go from the **Geometry** node to the **Material** node or a physics interface node in the Model Builder. In some situations this operation might fail.

As a result of these performance issues, the ECAD Import has its own extrude operation that automatically connects non matching planes. In one operation this functionality extrudes and connects all layers, so there is only one geometry object after the import. With only one object, it is easy to switch to the physics modes. Use this special extrude operation when using the grouping option **All**.

The special extrude operation is bound to certain rules that the 2D layout must fulfill. If the 2D layout does not comply with these rules, the operation might fail. Then switch to one of the other grouping options to import the geometry.

Importing ODB++(X) Files

If your ECAD software supports the ODB++(X) format it is recommended it is used as it usually gives the most efficient geometry model of the layout.

The ODB++ file format is a sophisticated format that handles most of the information needed to manufacture a PCB. Some of the information is not needed when importing the file and the program ignores such information during import.

ODB++ exists in two different format versions:

- A single XML file containing all information organized in a hierarchy of XML tags. This file format is usually referred to as ODB++(X), and it is the only format that can be imported into COMSOL Multiphysics.
- A directory structure with several files, each containing parts of information about the PCB. An entire PCB layout is often distributed as zipped or unzipped tar archives. This version is currently not possible to import.

The ODB++ import reads the layer list and the first step in the file. Multiple step files are not yet supported. From the first step it reads all the layer features and the board outline but currently skips all the package information.

EXTRACTING LAYER STACKUP

The import can read stackup information from the ODB++ file, such as thickness for metal layers and dielectric layers. It is quite common that the layer thickness is not included in the export from the ECAD program, so the layers only get a default thickness. The thickness can always be changed prior to import on the **Layers to import** table in the settings window for the ECAD import, so it is recommended that these values are checked before importing.

Importing GDS-II Files

The GDS-II file format is commonly used for mask layout production used in the manufacturing process of semiconductor devices and MEMS devices. The file is a

binary file, containing information about drawing units, geometry objects, and object drawing hierarchy. The drawing hierarchy is made up of a library of cell definitions, where each cell can be instantiated (drawn several times) with scaling, translation, mirroring, and rotation. It is also possible to repeat a cell as an array of drawn objects. This is very useful for mask layouts of integrated circuits, which often consist of millions of transistors. There are usually only a few transistor configurations present on the layout, and each transistor configuration only has to be defined once.

File Extension

The file extension of the GDS-II format is usually `.gds`, and the ECAD import requires it to be so, otherwise it cannot identify the file as a GDS-II file. If the file has a different extension, it must be changed to `.gds` before importing the file.

SUPPORTED FEATURES

There are several record types in a GDS file that are of no interest in a geometry import and these are ignored. There are also a few record types that actually could be imported as a geometry object, but are also ignored. One such example is the Text record, which produce a lot of mesh elements and is usually of no interest in a simulation. Below is a list of the supported record types.

- Boundary: a closed polyline object
- Box: a box object
- Path: a path with a thickness
- Sref: an instance of a cell that can be translated, rotated, scaled, and mirrored
- Aref: an n-by-m array of Sref objects
- Element: specification of a cell

3D IMPORT OF GDS-II FILES

The GDS-II format does not contain any information about layer thickness and layer position, so any such information has to be supplied by the user. When importing a GDS-II file with the ECAD import, it creates a table for all layers included in the file. In that table it is possible to specify a thickness for each layer and thereby get a 3D structure. This procedure has a few limitations regarding how the GDS layers are organized:

- One layer represents one position in height, so if the file contains two GDS layers that define two objects on the same height, the ECAD import still positions the layers with one layer on top of the other. Several GDS layers on the same height is common for semiconductor layouts, where the fabrication process includes

deposition followed by etching and then redepositing of a different layer. Such advanced process schemes cannot be automatically handled correctly by the ECAD import.

- With the grouping option **All**, objects on adjacent layers must not cross each other, because the original edge of the objects must be kept unchanged when two adjacent layers are merged to form the interface between them. You can get around this by selecting a different grouping option (see ECAD Import).
- Use the 3D GDS-II import with the ECAD import. The standard CAD import of COMSOL Multiphysics does not support pre-reading of the file, so it is not possible to specify any properties the layers (like thickness for example). The ECAD import always reads the file before displaying the import options.

The best way to solve any of these issues is to do the import with the grouping option **By layer**, and manually rearrange the layers by simple move operations so the elevation of the layers are correct. You can do etching by removing a layer from other objects, using the **Difference** button on the main toolbar or the **Difference** feature from the **Boolean Operations** submenu on the **Geometry** node's context menu.

Importing NETEX-G Files

The NETEX-G file format is a special format produced by the application NETEX-G by Artwork (www.artwork.com). NETEX-G can read Gerber and drill files that almost any ECAD software can export to because those formats are used when sending the layout to manufacturing. The output file is an ASCII file with a GDS-like structure, containing information about the layout of each layer, the layer thickness, vias, and dielectric layers. The geometry objects are defined and instantiated in the same way as in a GDS file; see Importing GDS-II Files for a more detailed description.

File Extension

The file extension of the NETEX-G format is not set, but the ECAD import requires it to be `.asc`, otherwise it cannot identify the file as a NETEX-G file. If the file has a different extension, change the name before importing it. Throughout the rest of this section, files of this type are referred to as a Netex file.

USING NETEX-G

This is a brief description of the main steps to produce a Netex file for import into COMSOL Multiphysics. For specific details see the NETEX-G user guide.

GERBER Layer Files

The first type of input files to NETEX-G is a collection of Gerber files, one for each layer. The ECAD software generates these files when the PCB layout is sent to manufacturing, but they can also be used for interfacing to other programs like COMSOL Multiphysics. The layer files do not contain any information about layer thickness, layer materials, dielectrics, and electrical connectivity (nets). Furthermore, a standard PCB layout usually consists of a large number of conductors, vias, and symbols printed in metal that are not important for a finite element simulation. With NETEX-G the size of the exported layout can be reduced in the following ways:

- Defining a region to include in the export. This region is drawn directly on a top view of the layout.
- Exclude entire layers from the layout.
- Selecting electrical nets to include in the export in addition to the selected region.
- It is also possible to let NETEX-G include nets in the proximity of the selected nets.

Because the Gerber layer files do not contain any physical information about the layer and dielectrics, this information must be specified in NETEX-G.

Some of these steps can also be done during import to COMSOL Multiphysics, for example, excluding layers from the import and changing thickness of the layers.

Drill Files

The connectivity between the layers is defined through drilled holes, known as *vias*. A via can go through the entire circuit board or just between certain layers. Most ECAD programs use the Excellon drill file format to specify the vias, which contains information about via diameter and position. Before generating the final output file from NETEX-G, it is necessary to convert all drill files to Gerber format and include them to the export project in NETEX-G. For each drill file, it is also necessary to specify between which layers the hole goes. Within NETEX-G a tool can be called that directly converts the Excellon drill format into Gerber. After the conversion, also specify the source and destination layers for the drill file.

NETEX-G Export Settings

To reduce the complexity of the output file it is recommended that vias are exported as circles and not as polygon chains. Although the arc recognition utility can detect these polygons, the former option is a bit more robust.

IMPORTING WIREBONDS

The Netex file can contain information about wirebonds or bond wires. Including wirebonds in the geometry often increases the problem size significantly. To get more control over the problem size, control the complexity of the imported wires.

Types of Wirebonds

The ECAD import can model the wirebond at three different complexity levels:

- As geometrical edges. This is the simplest form, which works well when the current in the wires is known.
- As solids with a square-shaped cross section. This cross section often produces fewer mesh elements than when using a circular cross section and is also easier for the geometry engine to analyze.
- As solids with a circular cross section.

Wirebonds Models

The Netex file format supports wirebonds models according to the JEDEC standard. It is possible to define the wirebond as a JEDEC3 or a JEDEC4 model. These models define the bond wire as 3- or 4-segment paths with user-supplied coordinates and elevations. In a Netex file the bond wire goes from a layer to a special die layer, representing the semiconductor die.

Important

Wirebonds are currently not supported with the grouping option set to **All**. Using this option ignores all wirebonds.

ECAD Import Options

ECAD IMPORT

Most PCB layout files mainly contain definitions of 2D objects. The Netex file also contains information about wirebonds. The ECAD import engine first creates the 2D objects for each layer, possibly grouped as one object. Then it extrudes all the objects in each layer according to the information in the file. GDS files contain no information about thickness, so a default value of 100 µm is used for all layers. The ECAD Import allows the layer thickness to be changed prior to import. Another alternative is to first import the objects into 2D and then manually extrude them to 3D.

Right-click the **Geometry** node to add an **Import** node. Under **Geometry import** in the **Import** section, decide the type of CAD file to import—**ECAD file (GDS/NETEX-G)** or **ECAD file (ODB++)**. Enter the path to the file or click **Browse** to locate the file to import. Before clicking the **Import** button consider the import options described below.

THE ECAD IMPORT OPTIONS

There are a number of settings that control how to treat the information in the layout file. The content of this section depends on the file type to be imported.

For GDS and NETEX-G files, enter a *net* name in the **Net to import (blank means top net)** field if you want to import a single electrical net beneath the top net in the hierarchy. Leave this field empty to import the top net (top cell). (In GDS files, the standard terminology is *cell* instead of net, but structurally they mean the same thing.)

The **Grouping of geometries** list specifies how the imported geometry objects are grouped in the final geometry. The choices for 3D import are:

- **All**. Groups all objects into one single object. This selection makes use of a more efficient extrude algorithm that extrudes and combines all layers directly. Because the import results in only one geometry object, COMSOL Multiphysics does not need to do a complicated analysis of several geometry objects.
- **By layer**. Groups all objects in one layer into one geometry object. The final geometry contains one object for each layer.
- **No grouping**. No grouping of objects is performed. This can be useful for debugging purposes when the other choices fail for some reason. This selection returns all the primitive objects found in the file, so objects with negative polarity are not drawn correctly.

The **Type of import** list specifies how to treat metal layers. The **Full 3D** option imports all metal layers with a thickness. Select the **Metal shell** options if you want to import all metal layers as an embedded boundary between dielectric regions.

For NETEX-G files, bond wires or wirebonds can be imported using three different complexity levels. Choose the level from the **Type of bond wires** list:

- **Edges**. The path of the bond wire is represented only as a geometrical edge. This option has the least complexity and does not produce a large number of mesh elements. There might be some limitations when using these edges in modeling.
- **Blocks**. The bond wire is modeled as a solid with a square cross section.
- **Cylinders**. Same as above but with a circular cross section.

Select the **Manual control of elevations** check box to manually position the layers in the *z* direction. This check box is enabled when **Grouping of geometries** is set to **By layer** or **No grouping**. When **Manual control of elevations** is not enabled, the *z* positions of the layers are calculated automatically from the layer **Thickness** values.

The layer information from the file appears in the **Layers to import** table. In addition to the layer **Name**, the table includes the following columns:

- The **Type** column. This column declares the type of layer. The import treats layers of different types differently. For example, a layer of type **Metal** converts to faces if the option **Type of import** is set to **Metal shell**. The **Outline** type uses a union of the objects in the selected layer as a PCB outline. For ODB++ files, the **Drill** type means that the objects in the layer define drilled via holes through the PCB. For NETEX-G files, the vias are defined within each metal and dielectric layer.
- The numbers in the **Thickness** column can be changed.

Important	The **Thickness** column is especially important when importing GDS files because that format does not contain any thickness information, so all layers get a default thickness that you probably want to change.

- The number in the **Elevation** column can be changed. The **Elevation** column controls the lower Z position of a layer. The **Elevation** column is only displayed when **Manual control of elevations** is enabled.
- The **Import** column. Clear the check box for layers that do not need to be imported.

Note	If the **Metal shells** import type is used, isolated boundaries cannot be imported if the import also includes another solid layer. Then two imports must be performed. The only exception to this rule is when the import results in only face objects.

In most electromagnetic simulations the material between the metal layers is important for the simulation result. For NETEX-G/GDS import, the **Import dielectric regions** check box controls if the import engine also includes the dielectric layers, which in most cases are the actual PCB materials. An ODB++ file usually has the outline of the PCB board defined in the file. If a NETEX-G file or a GDS file is imported, it is possible to define the PCB outline using left, right, top, and bottom margins for the dielectric material. They define the distance between the exterior of the PCB and the

bounding box of all metal layers. The **Import dielectric regions** check box is disabled when **Manual control of elevations** is enabled.

With the **Keep interior boundaries** check box cleared, the import removes all interior boundaries of the imported nets. This keeps the geometry complexity to a minimum and can also make the import more robust in some situations.

Clearing the **Ignore text objects** check box tells the importer to skip all objects in an ODB++ file that have the TEXT tag set. It is common that PCB layouts have text written in copper. Such objects increase the problem size and are usually of no interest in a physical simulation.

For NETEX-G/GDS import, other options that can significantly reduce the complexity of imported layouts are the recognition of arcs and straight lines. With the **Recognize arcs** set to **Automatic**, all polygon chains that represent arcs are identified and replaced with more efficient curve objects. With the fields appearing when setting this to **Manual**, the arc recognition can be fine tuned. The **Find straight lines** check box also controls whether to convert several polygon segments that lie on a single straight line into a single straight segment. This option uses the number in the **Minimum angle between segments** field to determine if a group of segments lies on the same straight line.

Geometry repair is controlled via the **Repair imported data** check box and the **Relative repair tolerance** field.

Meshing an Imported Geometry


See Also

Creating Meshes and Generating a 3D Swept Mesh in the *COMSOL Multiphysics User's Guide* and Convert in the *COMSOL Multiphysics Reference Guide*.

The imported geometry often consists of objects with very high aspect ratios, which are hard to mesh with a free tetrahedron mesh generator. As a result, it is often necessary to use interactive meshing of the imported geometry in a by-layer fashion.

The following section describes this procedure in general terms.

This procedure assumes that the top and bottom layers are metal layers. All metal layers can often be meshed using swept meshing, but dielectric layers usually cannot be meshed that way. Begin by meshing from the bottom or top layer, starting with a

boundary mesh. Then mesh layer by layer, where each metal layer gets a swept mesh, and each dielectric layer (with vias) gets a free mesh.

The dielectric layers cannot use a swept mesh because the source and target boundaries usually do not look the same. If there is a surrounding air domain it is usually not possible to use swept meshes for the metal layers either. Use tetrahedrons or convert the swept mesh to tetrahedrons before meshing the surrounding domain.

Troubleshooting ECAD Import

TUNING IMPORT SETTINGS

Delete Interior Edges

A complex layout produces a large number of faces that can be hard to render. A simple way to reduce the number of faces is to clear the **Keep interior boundaries** check box in the ECAD import options. This removes all faces internal to the nets within a layer.

Removing Features

Remove all features that are not important for the simulation. This is usually best to do before the import in NETEX-G or in the ECAD software. When importing with **Grouping of geometries** set to **None** it is possible to manually delete certain objects after import, but it is recommended to do this only for relatively simple geometries.

PROBLEMS WHEN EXTRUDING LAYERS

Most ECAD or EDA programs support design rule checks (DRC), which test the entire layout and check that all features (vias, conductors, and components) are separated according to certain rules. With such checks the layout is free from overlapping vias and conductors touching other conductors or vias. This also ensures that the special extrude functionality of the ECAD import works properly. If the file contains such design-rule violations, the extrude might fail and throw an error message stating that it could not handle the topology of the layout.

The best approach to handle such problems is to perform a DRC with your ECAD software and produce new layout files. If this is not possible, import the layout in 2D and try to identify the problematic features. They can either be in a single layer or at the interface between two adjacent layers. When identified, it is possible to remove them manually using a text editor if a NETEX-G file or an ODB++ file is being imported. It can be hard to find a certain feature, but use either the coordinate or the net information to find it. The GDS format is a binary file format so it is very difficult to edit the file manually.

PROBLEMS WITH SEVERAL GEOMETRY OBJECTS

If the special extrude functionality is not used, you get several geometry objects, for example, one for each layer if **By layer** is selected from the **Grouping of geometries** list. After a CAD import COMSOL Multiphysics is in the Geometry branch of the model tree. When you continue to the **Materials** branch if the model tree or to a physics interface node or the **Mesh** branch, the program tries to combine all the objects into one geometry, and this operation might fail if the objects are very complex and have high aspect rations. Resolve this either by trying the option **All** in the **Grouping of geometries** list. This creates one combined geometry object by using the special extrude functionality, and with only one object this.

Another possibility is to use assemblies, because then COMSOL Multiphysics does not have to combine the objects (parts). This is controlled by the **Finalize** node in the Geometry branch of the model tree. When using an assembly, use identity pairs to connect the interfaces between the layers.

As a final option, do not import the dielectric layers. The import then leaves isolated metal layers that have to connect with coupling variables.

4

The Electric Field Interfaces

This chapter summarizes the functionality of the Electric Field interfaces, which are found under the **AC/DC** branch () in the **Model Wizard**. The AC/DC Module enhances the Electrostatics and Electric Currents interfaces included with the basic COMSOL Multiphysics license.

In this chapter:

The Electrostatics Interface

The **Electrostatics** interface ( ), found under the **AC/DC** branch ( ) in the **Model Wizard**, has the equations, boundary conditions, and space charges for modeling electrostatic fields, solving for the electric potential.

Charge Conservation is the main feature, which adds the equation for the electric potential and has a settings window for defining the constitutive relation for the electric displacement field and its associated properties such as the relative permittivity.

When this interface is added, these default nodes are also added to the **Model Builder**—**Charge Conservation**, **Zero Charge** (default boundary condition), and **Initial Values**. Right-click the **Electrostatics** node to add other features that implement, for example, boundary conditions and space charges.


Model

Electric Sensor: Model Library path **COMSOL_Multiphysics/Electromagnetics/electric_sensor**

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `es`.

DOMAIN SELECTION

The default setting is to include **All domains** in the model to define the electric potential and the equations that describe the potential field for dielectrics. To choose specific domains, select **Manual** from the **Selection** list.

THICKNESS

1D	Enter a default value for the **Cross-section area** A (SI unit: m^2). The default value of 1 is typically not representative for a thin domain. Instead it describes a unit thickness that makes the 1D equation identical to the equation used for 3D models. See also Change Cross-Section.

 *2D*	Enter a default value for the **Out-of-plane thickness** d (SI unit: m). The default value of 1 is typically not representative for a thin dielectric medium, for example. Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D models. See also Change Thickness (Out-of-Plane).

SWEEP SETTINGS

Select the **Activate terminal sweep** check box to switch on the sweep and invoke a parametric sweep over the terminals. Enter a **Sweep parameter name** to assign a specific name to the variable that controls the terminal number solved for during the sweep. The **Sweep parameter name** must also be declared as a model parameter. The default is `PortName`.

In the **Model Builder**, right-click **Global Definitions** and choose **Parameters** and enter the chosen name and assign to it a temporary **Expression** of unity into the **Parameters** table.

 *Important*	Only a temporary expression needs to be entered at this stage. During the analysis process, the solver assigns a proper value to this parameter. The generated lumped parameters are in the form of capacitance matrix elements. The terminal settings must consistently be of either fixed voltage or fixed charge type.

DISCRETIZATION

To display this section, select click the **Show** button ( ) and select **Discretization**. Select an element order for the **Electric potential**—**Linear**, **Quadratic** (the default), **Cubic**, **Quartic**, or (in 2D only) **Quintic**. Specify the **Value type when using splitting of complex variables**—**Real** or **Complex** (the default).

DEPENDENT VARIABLES

The dependent variable (field variable) is for the **Electric potential** *V*. The name can be changed but the names of fields and dependent variables must be unique within a model.


See Also

- The Model Builder Show and Hide Physics Options
- Domain, Boundary, Edge, Point, and Pair Conditions for the Electrostatics Interface
- Theory for the Electrostatics Interface

Domain, Boundary, Edge, Point, and Pair Conditions for the Electrostatics Interface

The Electrostatics Interface has these domain, boundary, edge, point, and pair features available.

About the Boundary Conditions

The relevant interface condition at interfaces between different media is

$$\mathbf{n}_2 \cdot (\mathbf{D}_1 - \mathbf{D}_2) = \rho_s$$

In the absence of surface charges, this condition is fulfilled by the natural boundary condition

$$\mathbf{n} \cdot [(\varepsilon_0 \nabla V - \mathbf{P})_1 - (\varepsilon_0 \nabla V - \mathbf{P})_2] = -\mathbf{n} \cdot (\mathbf{D}_1 - \mathbf{D}_2) = 0$$

Available Features

These features are available for this interface and listed in alphabetical order. Also see Table 4-1 for a list of interior and exterior boundary conditions, including edge, point, and pair availability.

- Change Cross-Section
- Change Thickness (Out-of-Plane)
- Charge Conservation
- Dielectric Shielding
- Distributed Capacitance
- Electric Displacement Field
- Electric Potential

- Electrostatic Point Dipole
- External Surface Charge Accumulation
- Floating Potential
- Force Calculation
- Ground
- Initial Values
- Line Charge
- Line Charge (on Axis)
- Line Charge (Out-of-Plane)
- Periodic Condition
- Point Charge
- Point Charge (on Axis)
- Space Charge Density
- Surface Charge Density
- Terminal
- Thin Low Permittivity Gap
- Zero Charge (the default boundary condition)

Table 4-1 lists the interior and exterior boundaries available with this interface. It also includes edge, point, and pair availability.

TABLE 4-1: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS (INCLUDING EDGE, POINT, AND PAIR AVAILABILITY) FOR THE ELECTROSTATICS INTERFACE

FEATURE	INTERIOR	EXTERIOR	ALSO AVAILABLE FOR
Change Cross-Section	x	x	pairs
Change Thickness (Out-of-Plane)	x	x	pairs
Dielectric Shielding	x	x	pairs
Distributed Capacitance	x	x	pairs
Electric Displacement Field	x	x	pairs
Electric Potential	x	x	edges, points, and pairs
External Surface Charge Accumulation		x	pairs
Floating Potential	x	x	pairs
Ground	x	x	edges, points, and pairs
Periodic Condition		x	not applicable

TABLE 4-1: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS (INCLUDING EDGE, POINT, AND PAIR AVAILABILITY) FOR THE ELECTROSTATICS INTERFACE

FEATURE	INTERIOR	EXTERIOR	ALSO AVAILABLE FOR
Surface Charge Density	x	x	pairs
Terminal	x	x	not applicable
Thin Low Permittivity Gap	x		not applicable
Zero Charge (the default)	x	x	pairs


1D Axi


2D Axi

For axisymmetric models, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an **Axial Symmetry** feature to the model that is valid on the axial symmetry boundaries only. There are also **Line Charge (on Axis)** and **Point Charge (on Axis)** features available.


See Also

In the *COMSOL Multiphysics User's Guide*:

- Continuity on Interior Boundaries
- Identity and Contact Pairs
- Specifying Boundary Conditions for Identity Pairs


Tip

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

Charge Conservation

The **Charge Conservation** node adds the equations for charge conservation according to Gauss' law for the electric displacement field. It provides an interface for defining the constitutive relation and its associated properties such as the relative permittivity.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the electric potential and the equation based on Gauss' law that describes the potential field.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty.

MATERIAL TYPE

Select a **Material type**—**Solid**, **Non-solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

ELECTRIC FIELD

Select a **Constitutive relation** to describe the macroscopic properties of the medium (relating the electric displacement $\mathbf{D}$ with the electric field $\mathbf{E}$) and the applicable material properties, such as the relative permittivity.

Select a **Constitutive relation** from the list—Select **Relative permittivity** to use the constitutive relation $\mathbf{D} = \varepsilon_0\varepsilon_r\mathbf{E}$ (the default). Select **Polarization** to use the constitutive relation $\mathbf{D} = \varepsilon_0\mathbf{E} + \mathbf{P}$. Select **Remanent electric displacement** to use constitutive relation $\mathbf{D} = \varepsilon_0\varepsilon_r\mathbf{E} + \mathbf{D}_r$, where $\mathbf{D}_r$ is the remanent displacement (the displacement when no electric field is present).

- If **Relative permittivity** is selected, the default is to take the **Relative permittivity** (ε_r) values **From material**. If **User defined** is selected, select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** and enter values or expressions in the field or matrix. Select **Porous media** to then be able to right-click the **Charge Conservation** node to add a Porous Media subnode.
- If **Polarization** is selected, enter the components based on space dimension for the **Polarization** vector $\mathbf{P}$ (SI unit: C/m^2).
- If **Remanent electric displacement** is selected, the default is to take the **Relative permittivity** (ε_r) values **From material**. If **User defined** is selected, select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** and enter values or expressions in the field or matrix. Then enter the components based on space dimension for the **Remanent electric displacement** $\mathbf{D}_r$ (SI unit: C/m^2). Select **Porous media** to then be able to right-click the **Charge Conservation** node to add a Porous Media subnode.

Space Charge Density

The **Space Charge Density** node adds a space charge density ρ, which appears on the right-hand side of the equation that the interface defines.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define a space charge density.

SPACE CHARGE DENSITY

Enter a value or expression for the **Space charge density** ρ_q (SI unit: $\mathrm{C/m^3}$).

Force Calculation

Use the **Force Calculation** node to define globally available force and torque variables for the selected domains.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define a force calculation.

FORCE CALCULATION

Enter a **Force name**, which is then appended to global variables. The method used to compute forces and torques is integration of the Maxwell's stress tensor over the exterior surfaces of the set of domains. This feature also gives access to the normal component of the Maxwell stress tensor on the external surfaces.

Note

For the **Magnetic and Electric Fields** and **Magnetic Fields** interfaces, the force calculation includes both electric and magnetic forces.

Enter a direction vector for the **Torque axis** $\mathbf{r}_{\mathrm{ax}}$ and coordinates for the **Torque rotation point** $\mathbf{r}_0$. A torque calculation about a given point (**Torque rotation point**) is made and defined as a global vector variable, `es.T_<force name><component>`. The resulting torque component parallel to the given **Torque axis** is given as a global variable, typically `es.Tax_<force name>`.

Initial Values

The **Initial Values** node adds an initial value for the electric potential V that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define an initial value.

INITIAL VALUES

Enter a value or expression for the initial value of the **Electric potential** V. The default value is 0 V.

Ground

The **Ground** node is the default boundary condition and implements ground as the boundary condition $V = 0$.

Ground means that there is a zero potential on the boundary. This boundary condition is also applicable at symmetry boundaries where the potential is known to be antisymmetric with respect to the boundary.

For some interfaces, also select additional **Ground** features from the **Edges** (3D models) or **Points** (2D and 3D models) submenus. For 2D axisymmetric models, it can be applied on the Symmetry axis.

BOUNDARY, EDGE, OR POINT SELECTION

From the **Selection** list, choose the geometric entity (boundaries, edges, or points) to apply a ground (zero potential) boundary condition.

Caution

Beware that constraining the potential on edges or points in 3D or on points in 2D usually yields a current outflow that is mesh dependent.

PAIR SELECTION

If **Ground** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

Electric Potential

The **Electric Potential** node provides an electric potential V_0 as the boundary condition $V = V_0$.

Because the electric potential is being solved for in the interface, the value of the potential is typically defined at some part of the geometry. For some interfaces, also select additional **Electric Potential** features from the **Edges** (3D models) or **Points** (2D and 3D models) submenus. For 2D axisymmetric models, it can be applied on the Symmetry axis.

BOUNDARY, EDGE, OR POINT SELECTION

From the **Selection** list, choose the geometric entity (boundaries, edges, or points) to apply an electric potential as the boundary condition.


Caution

Beware that constraining the potential on edges or points in 3D or on points in 2D usually yields a current outflow that is mesh dependent.

PAIR SELECTION

If **Electric Potential** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

ELECTRIC POTENTIAL

Enter the value or expression for the **Electric potential** V_0 (SI unit: V).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

Surface Charge Density

The **Surface Charge Density** node provides the following surface-charge boundary condition for exterior boundaries (left) and interior boundaries (right):

$$-\mathbf{n}\cdot\mathbf{D} = \rho_s, \qquad \mathbf{n}\cdot(\mathbf{D}_1 - \mathbf{D}_2) = \rho_s$$

Specify the surface charge density ρ_s at an outer boundary or at an interior boundary between two nonconducting media.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply a surface charge density.

PAIR SELECTION

If **Surface Charge Density** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

SURFACE CHARGE DENSITY

Enter the value or expression for the **Surface charge density** ρ_s (SI unit: C/m^2).

External Surface Charge Accumulation

The **External Surface Charge Accumulation** node implements the boundary condition

$$-\mathbf{n}\cdot\mathbf{D} = \rho_s$$

where ρ_s is the solution of the following distributed ODE on the boundary:

$$\frac{d\rho_s}{dt} = \mathbf{n}\cdot\mathbf{J}_i + \mathbf{n}\cdot\mathbf{J}_e$$

where $\mathbf{n}\cdot\mathbf{J}_i$ is the normal component of the total ion current density on the wall and $\mathbf{n}\cdot\mathbf{J}_e$ is the normal component of the total electron current density on the wall, which are feature inputs.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply a surface charge accumulation.

PAIR SELECTION

If **External Surface Charge Accumulation** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

EXTERNAL SURFACE CHARGE ACCUMULATION

Enter values or expressions for the **Normal ion current density** $\mathbf{n} \cdot J_i$ (SI unit: $\mathrm{A/m^2}$) and the **Normal electron current density** $\mathbf{n} \cdot J_e$ (SI unit: $\mathrm{A/m^2}$).

Dielectric Shielding

The **Dielectric Shielding** node adds dielectric shielding as a boundary condition. It describes a thin layer with thickness d_s and a bulk relative permittivity; ε_r that shields the electric field:

$$\mathbf{n} \cdot \mathbf{D} = -\nabla_\mathrm{t} \cdot \varepsilon_0 \varepsilon_{\mathrm{r}s} d_\mathrm{s} \nabla_\mathrm{t} V$$

Use this boundary condition when approximating a thin domain with a boundary to reduce the number of mesh elements.

BOUNDARY OR EDGE SELECTION

From the **Selection** list, choose the geometric entity (boundaries or edges) to apply a dielectric shielding as the condition.

PAIR SELECTION

If **Dielectric Shielding** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

MATERIAL TYPE

Select a **Material type**—**Solid**, **Non-solid**, or **From material**.

ELECTRIC SHIELDING

The default is to take the **Relative permittivity** ε_r (unitless) values **From material**. It takes it from the adjacent domains if not explicitly defined. If **User defined** is selected, select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** and enter values or expressions in the field or matrix.

Enter a **Surface thickness** d_s (SI unit: m) of the shielding.

Terminal

The **Terminal** node provides a boundary condition for connection to external circuits, to transmission lines, or with a specified voltage or current. By specifying zero current, a floating potential condition is obtained.


See Also

Lumped Parameters

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to model as terminals connected to external circuits or an external current or voltage.


Note

For the **Electric Currents, Shell** interface, select edges (3D) or points (2D) instead of boundaries.

PAIR SELECTION

If **Terminal** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

TERMINAL

Specify the terminal's properties. To indicate which boundaries that belong to the same terminal, enter the same name in the **Terminal name** field. The **Terminal name** should be numeric for sweeps to work properly.

Select a **Terminal type**—**Voltage**, **Current**, **Circuit**, or **Terminated**. Select:

- **Voltage** to enter an electric potential V_0 (SI unit: V). The default is 1.
- **Charge** to enter current I_0 (SI unit: C). The default is zero current for an open circuit.

- **Circuit** to specify a terminal connected to an external circuit.

Important	The **Circuit** type should not be used for lumped parameter calculations. For the terminal, also enter the value of the electric potential or current/charge used if required. If zero is entered, the terminal acts as a floating electrode.

- **Terminated** to connect the terminal to an impedance that may represent a load or a transmission line. When **Terminated** is selected, the scattering parameters (S-parameters) are computed.
  - Enter a **Terminal power** P_0 (SI unit: W) to specify the input power at this terminal from the transmission line. This excitation can be toggled using a port sweep.
  - Select an option from the **Characteristic impedance from** list to define the value of the impedance—**Physics interface** or **User defined**. If **Physics interface** is selected, the **Reference impedance** Z_{ref} defined on the physics interface settings window under Sweep Settings is used. If **User defined** is selected, enter a specific impedance Z_{ref} (SI unit: Ω) for this terminal. The default is 50 Ω.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

Floating Potential

See Also	Lumped Parameters

Note	For the **Electric Currents, Shell** interface, select edges (3D) or points (2D) instead of boundaries.

Important	The **Circuit** type should not be used for lumped parameter calculations. For the terminal, also enter the value of the electric potential or current/charge used if required. If zero is entered, the terminal acts as a floating electrode.

The **Floating Potential** node is used when modeling a metallic electrode at floating potential. The electrode may have a charge Q_0 deposited on it. For circuit connections use the **Terminal** feature instead.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define the floating electrode.

PAIR SELECTION

If **Floating Potential** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

FLOATING POTENTIAL

Specify a an optionally non zero **Charge** Q_0 (SI unit: C).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

Electric Displacement Field

The **Electric Displacement Field** node adds the following electric-displacement boundary condition:

$$\mathbf{n} \cdot \mathbf{D} = \mathbf{n} \cdot \mathbf{D}_0$$

It specifies the normal component of the electric displacement field at a boundary.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to use the normal component of the displacement field as the boundary condition.

PAIR SELECTION

If **Displacement Field** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

ELECTRIC DISPLACEMENT FIELD

Enter the coordinates of the **Boundary electric displacement field** $\mathbf{D}_0$ (SI unit: C/m^2).

Distributed Capacitance

The **Distributed Capacitance** node adds a distributed capacitance boundary condition according to the following equations for exterior boundaries (left) and interior boundaries (right):

$$-\mathbf{n}\cdot\mathbf{D} = \varepsilon_0\varepsilon_r\frac{V_{ref}-V}{d_s} \qquad \mathbf{n}\cdot(\mathbf{D}_1-\mathbf{D}_2) = \varepsilon_0\varepsilon_r\frac{V_{ref}-V}{d_s}$$


Tip

Use this boundary condition to model a thin sheet or film of a dielectric material. The sheet has the relative permittivity ε_r and the surface thickness d_s, and it is connected to the reference potential V_{ref}.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply a distributed capacitance.

PAIR SELECTION

If **Distributed Capacitance** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

MATERIAL TYPE

Select a **Material type**—**Solid**, **Non-solid**, or **From material**.

DISTRIBUTED CAPACITANCE

Enter the values or expressions for **Relative permittivity** ε_r, **Surface thickness** d_s (SI unit: m), and **Reference potential** V_{ref} (SI unit: V). The default value for the surface thickness is 10^{-3} m (1 mm).

Periodic Condition

The **Periodic Condition** node defines periodicity or antiperiodicity between two boundaries. If required, activate periodic conditions on more than two boundaries, in which case the Periodic Condition tries to identify two separate surfaces that can each consist of several connected boundaries.

Tip

For more complex geometries it might be necessary to use the **Destination Selection** subnode. With this subnode the boundaries which constitute the source and destination surfaces can be manually specified. To add the subnode, right-click the **Periodic Condition** node and select **Destination Selection**.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply a periodic condition.

PERIODIC CONDITION

Select a **Type of periodicity**—**Continuity** or **Antiperiodicity**.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

See Also

In the *COMSOL Multiphysics User's Guide*:

- Periodic Condition
- Destination Selection
- Using Periodic Boundary Conditions
- Periodic Boundary Condition Example

Zero Charge

The **Zero Charge** node adds the condition that there is zero charge on the boundary so that $\mathbf{n} \cdot \mathbf{D} = 0$. This boundary condition is also applicable at symmetry boundaries where the potential is known to be symmetric with respect to the boundary. This is the default boundary condition at exterior boundaries. At interior boundaries, it means

that no displacement field can penetrate the boundary and that the electric potential is discontinuous across the boundary.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply a zero charge condition.

PAIR SELECTION

If **Zero Charge** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

Thin Low Permittivity Gap

Use the **Thin Low Permittivity Gap** node

$$\mathbf{n} \cdot \mathbf{D}_1 = \frac{\varepsilon_0 \varepsilon_r}{d_s}(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{D}_2 = \frac{\varepsilon_0 \varepsilon_r}{d_s}(V_2 - V_1)$$

to model a thin gap of a material with a small permittivity compared to the adjacent domains. The layer has the thickness d_s and the relative permittivity ε_r. The indices 1 and 2 refer to the two sides of the boundary.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply a thin low permittivity gap condition.

MATERIAL TYPE

Select a **Material type**—**Solid**, **Non-solid**, or **From material**.

PAIR SELECTION

When **Thin Low Permittivity Gap** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

THIN LOW PERMITTIVITY GAP

The default is to take the **Relative permittivity** (ε_r) values **From material**. Select **User defined** to enter a different value or expression.

Enter a **Surface thickness** d_s (SI unit: m).

Line Charge


3D

For 3D models, use the **Line Charge** node to specify line charges along the edges of a geometry.

EDGE SELECTION

From the **Selection** list, choose the edges to add a line charge.


Caution

Beware that constraining the potential on edges usually yields a current outflow that is mesh dependent.

LINE CHARGE

Enter a value or expression to apply a **Line charge** Q_L (SI unit: C/m). This source represents electric charge per unit length.


See Also

- Line Charge (on Axis)
- Line Charge (Out-of-Plane)

Line Charge (on Axis)


2D Axi

For 2D axisymmetric models, use the **Line Charge (on Axis)** node to specify line charges along the symmetry axis.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries on axis to add a line charge.

LINE CHARGE (ON AXIS)

Enter a value or expression to apply a **Line charge** Q_L (SI unit: C/m). This source represents electric charge per unit length.


See Also

- Line Charge
- Line Charge (Out-of-Plane)

Line Charge (Out-of-Plane)


2D


2D Axi

For 2D and 2D axisymmetric models, points are selected and this is the same as a line out-of-plane.

Use the **Line Charge (Out-of-Plane)** node to specify line charges along the points of a geometry for 2D and 2D axisymmetric models.

POINT SELECTION

From the **Selection** list, choose the points to add a line charge.


Caution

Beware that constraining the potential on points usually yields a current outflow that is mesh dependent.

LINE CHARGE (OUT-OF-PLANE)

Enter a value or expression to apply a **Line charge** Q_L (SI unit: C/m). This source represents electric charge per unit length.


See Also

- Line Charge
- Line Charge (on Axis)

Point Charge


3D

The **Point Charge** node adds a point source to 3D models. The point charge represents an electric displacement field flowing out of the point.

POINT SELECTION

From the **Selection** list, choose the points to add a point charge.


Caution

Beware that constraining the potential on points usually yields a current outflow that is mesh dependent.

POINT CHARGE

Enter a value or expression to apply a **Point charge** Q_P (SI unit: C) to points. This source represents an electric displacement field flowing out of the point.


See Also

- Point Charge (on Axis)
- Line Charge (Out-of-Plane)

Point Charge (on Axis)


2D Axi

The **Point Charge (on Axis)** node adds a point source to 2D axisymmetric models. The point charge represents an electric displacement field flowing out of the point.

POINT SELECTION

From the **Selection** list, choose the points to add a point charge.


Caution

Beware that constraining the potential on points usually yields a current outflow that is mesh dependent.

POINT CHARGE (ON AXIS)

Enter a value or expression to apply a **Point charge** Q_P (SI unit: C) to points on axis. This source represents an electric displacement field flowing out of the point.


See Also

- Point Charge
- Line Charge (Out-of-Plane)

Change Cross-Section

1D

This feature is available with 1D models. This setting overrides the global **Thickness** setting made in any interface that uses this feature.

Use the **Change Cross-Section** feature to set the cross-section area for specific geometric entities.

DOMAIN OR BOUNDARY SELECTION

From the **Selection** list, choose the geometric entity (domains or boundaries) to define the change cross-section.

PAIR SELECTION

When **Change Cross-Section** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

CHANGE CROSS-SECTION

Enter a value or expression for the **Cross-section area** A (SI unit: m^2). The default value of 1 unit length is typically not representative for a thin domain. Instead it describes a

unit thickness that makes the 1D equation identical to the equation used for 3D models.


See Also

For 2D models, see Change Thickness (Out-of-Plane).

Change Thickness (Out-of-Plane)


2D

This feature is available for 2D models. This setting overrides the global **Thickness** setting made in any interface that uses this feature.

Use the **Change Thickness (Out-of-Plane)** feature to set the out-of-plane thickness for specific geometric entities.

DOMAIN OR BOUNDARY SELECTION

From the **Selection** list, choose the geometric entity (domains or boundaries) to define the change thickness.

PAIR SELECTION

When **Change Thickness (Out-of-Plane)** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

CHANGE THICKNESS (OUT-OF-PLANE)

Enter a value or expression for the **Out-of-plane thickness** d (SI unit: m). The default value of 1 unit length is typically not representative for a thin domain. Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D models.


See Also

For 1D models, see Change Cross-Section.

Infinite Elements

Note	For information about this feature, see About Infinite Element Domains and Perfectly Matched Layers in the *COMSOL Multiphysics User's Guide*.

Electrostatic Point Dipole

2D 3D	Add an **Electrostatic Point Dipole** node to 2D and 3D models.

The **Electrostatic Point Dipole** represents the limiting case of zero separation distance between two equally strong point sources of opposing signs while maintaining the product between separation distance and source strength at a fixed value (p). The dipole moment is a vector entity with positive direction from the negative charge to the positive one.

POINT SELECTION

From the **Selection** list, choose the points (also on axis) to add an electrostatic point dipole.

ELECTROSTATIC POINT DIPOLE

Select a **Dipole specification**—**Magnitude and direction** (the default) or **Electric dipole moment**.

- If **Magnitude and direction** is selected, enter coordinates for the **Electric dipole moment direction** $\mathbf{n}_p$ and the **Electric dipole moment, magnitude** p (SI unit: C·m).
- If **Dipole moment** is selected, enter coordinates for the **Electric dipole moment** $\mathbf{p}$ (SI unit: C·m).

The Electric Currents Interface

The **Electric Currents** interface ( ), found under the **AC/DC** branch ( ) in the **Model Wizard**, has the equations, boundary conditions, and current sources for modeling electric currents in conductive media, solving for the electric potential.

Current Conservation is the main feature, which adds the equation for the electric potential and provides a settings window for defining the electrical conductivity as well as the constitutive relation for the electric displacement field and its associated material properties such as the relative permittivity.

When this interface is added, these default nodes are also added to the **Model Builder**—**Current Conservation**, **Electric Insulation** (the default boundary condition), and **Initial Values**. Right-click the **Electric Currents** node to add other features that implement, for example, boundary conditions and current sources.

Model

Pacemaker Electrode: Model Library path **COMSOL_Multiphysics/Electromagnetics/pacemaker_electrode**

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `ec`.

DOMAIN SELECTION

The default setting is to include **All domains** in the model to define the electric potential and the equations that describe the potential field for conductive media. To choose specific domains, select **Manual** from the **Selection** list.

THICKNESS (1D)/ OUT-OF-PLANE THICKNESS (2D)

1D	Enter a default value for the **Cross-section area** A (SI unit: m^2). The default value of 1 is typically not representative for a thin domain. Instead it describes a unit thickness that makes the 1D equation identical to the equation used for 3D models. See also Change Cross-Section (described for the **Electrostatics** interface).

 *2D*	Enter a default value for the **Out-of-plane thickness** d (SI unit: m) (see Equation 4-1). The default value of 1 is typically not representative for a thin dielectric medium, for example. Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D models. See also Change Thickness (Out-of-Plane) (described for the **Electrostatics** interface).

TERMINAL SWEEP SETTINGS

Enter a **Reference impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω.

Select the **Activate terminal sweep** check box to switch on the sweep and invoke a parametric sweep over the terminals. Enter a **Sweep parameter name** to assign a specific name to the variable that controls the terminal number solved for during the sweep. The default is `PortName`. The generated lumped parameters are in the form of capacitance matrix elements. The terminal settings must consistently be of either fixed voltage or fixed charge type.

The lumped parameters are subject to **Touchstone file export**. Enter a file path or **Browse** for a file. Select an **Output format** for the Touchstone export—**Magnitude angle** (the default), **Magnitude (dB) angle**, or **Real imaginary**. Select a **Parameter to export**—**Z** (the default), **Y**, or **S**.

 *See Also*	Lumped Parameters

DEPENDENT VARIABLES

The dependent variable (field variable) is for the **Electric potential** V. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. Select an **Electric potential**—**Linear**, **Quadratic** (the default), **Cubic**, **Quartic**, or (in 2D only) **Quintic**. Specify the **Value type when using splitting of complex variables**—**Real** or **Complex** (the default).

See Also

- The Model Builder Show and Hide Physics Options
- Domain, Boundary, Edge, Point, and Pair Features for the Electric Currents Interface
- Theory for the Electric Currents Interface

Domain, Boundary, Edge, Point, and Pair Features for the Electric Currents Interface

The Electric Currents Interface has these domain, boundary, edge, point, and pair conditions available.

About the Boundary Conditions

The exterior and interior boundary conditions listed in Table 4-1 are available. The relevant interface condition at interfaces between different media and interior boundaries is continuity; that is,

$$\mathbf{n}_2 \cdot (\mathbf{J}_1 - \mathbf{J}_2) = 0$$

which is the natural boundary condition.

Available Features

These features and subfeatures are available for this interface and listed in alphabetical order. Also see Table 4-1 for a list of interior and exterior boundary conditions, including edge, point, and pair availability.

- Archie's Law
- Boundary Current Source

- Contact Impedance
- Current Conservation
- Current Source
- Distributed Impedance
- Electric Insulation
- Electric Point Dipole
- Electric Point Dipole (on Axis)
- Electric Shielding
- External Current Density
- Floating Potential
- Initial Values
- Line Current Source
- Line Current Source (on Axis)
- Normal Current Density
- Point Current Source
- Porous Media
- Sector Symmetry

These features are described for the **Electrostatics** interface:

- Change Cross-Section
- Change Thickness (Out-of-Plane)
- Electric Potential
- Force Calculation
- Ground
- Periodic Condition
- Terminal


See Also

In the *COMSOL Multiphysics User's Guide*:

- Continuity on Interior Boundaries
- Identity and Contact Pairs
- Specifying Boundary Conditions for Identity Pairs


1D Axi


2D Axi

For axisymmetric models, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an **Axial Symmetry** node to the model that is valid on the axial symmetry boundaries only.


Tip

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

Table 4-1 lists the interior and exterior boundaries available with this interface. It also includes edge, point, and pair availability.

TABLE 4-2: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS (INCLUDING EDGE, POINT, AND PAIR AVAILABILITY) FOR THE ELECTRIC CURRENTS INTERFACE

FEATURE	INTERIOR	EXTERIOR	ALSO AVAILABLE FOR
Boundary Current Source	x		pairs
Contact Impedance	x		pairs
Distributed Impedance	x	x	not applicable
Electric Insulation	x	x	pairs
Electric Potential	x	x	edges, points, and pairs
Electric Shielding	x	x	pairs
Floating Potential	x	x	pairs
Ground	x	x	edges, points, and pairs
Normal Current Density		x	not applicable
Periodic Condition		x	not applicable
Terminal	x	x	pairs

Current Conservation

The **Current Conservation** node adds the continuity equation for the electrical potential and provides an interface for defining the electric conductivity as well as the constitutive relation and the relative permittivity for the displacement current.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the electric potential and the continuity equation that describes the potential field.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty. If a linear temperature relation is added for the conductivity, then the source for the temperature T can be defined. From the **Temperature** list, select an existing temperature variable (from another physics interface) if available, or select **User defined** to define a value or expression for the temperature (SI unit: K) in the field that appears underneath the list.

MATERIAL TYPE

Select a **Material type**—**Solid** (the default), **Non-solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

CONDUCTION CURRENT

By default, the **Electrical conductivity** σ (SI unit: S/m) for the media is defined **From material**. Or select **User defined**, **Linearized resistivity**, **Archie's law**, or **Porous media**.

- If **User defined** is selected, select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** depending on the characteristics of the electrical conductivity, and then enter values or expressions in the field or matrix.


Note

If another type of temperature dependence is used other than a linear temperature relation, enter any expression for the conductivity as a function of temperature.

- Select **Linearized resistivity** for a temperature-dependent conductivity (this occurs in, for example, Joule heating, and is also called resistive heating). The equation describing the conductivity:

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

where ρ_0 is the resistivity at the reference temperature T_0, and α is the temperature coefficient of resistance, which describes how the resistivity varies with temperature.

- The default **Reference temperature** T_{ref} (SI unit: K), **Resistivity temperature coefficient** α (SI unit: 1/K), and **Reference resistivity** ρ_0 (SI unit: Ωm) are taken **From material**, which means that the values are taken from the domain material.


Note

Only certain material models (see Materials) support the **Linearized resistivity**.

- To specify other values for any of these properties, select **User defined** from the list and then enter a value or expression. T is the current temperature, which can be a value that is specified as a model input or the temperature from a heat transfer interface. The definition of the temperature field appears in the **Model Inputs** section.

Porous Media

When **Porous media** is selected, right-click to add a Porous Media subnode.

Archie's Law

When **Archie's law** is selected, right-click to add an Archie's Law subnode.

ELECTRIC FIELD


Note

See **Electric Field** as described for the Charge Conservation node for the Electrostatics interface.

Floating Potential

The **Floating Potential** node is useful when modeling a metallic electrode at floating potential. The electrode may have a charge Q_0 deposited on it. For circuit connections use the **Terminal** feature instead.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define the floating electrode.

PAIR SELECTION

If **Floating Potential** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

FLOATING POTENTIAL

Specify a an optionally non zero **Charge** Q_0 (SI unit: C).


Note

For the **Electric Currents** and **Magnetic and Electric Fields** interfaces, enter a **Terminal current** I_0 (SI unit: A). Specify zero current for a disconnected electrode.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

Archie's Law


Note

This subfeature is available only when Archie's law is selected as the **Electric conductivity** material parameter in the parent feature (for example, the **Current Conservation** node). Then right-click the Current Conservation node to add this subnode.

Use the **Archie's Law** subnode to provide an electrical conductivity computed using Archie's Law. This subnode can be used to model nonconductive porous media saturated (or variably saturated) by conductive liquids, using the relation:

$$\sigma = s_L^n \varepsilon_p^m \sigma_L$$


See Also

Archie's Law Theory

DOMAIN SELECTION

From the **Selection** list, choose the domains to define Archie's law.

MATERIAL TYPE

Select a **Material type**—**Solid**, **Non-solid**, or **From material**.

CONDUCTION CURRENT

By default, the **Electrical conductivity** σ_L (SI unit: S/m) for the fluid is defined **From material**. This uses the value of the conductivity of the material domain.

If **User defined** is selected, enter a value or expression. If another type of temperature dependence is used other than a linear temperature relation, enter any expression for the conductivity as a function of temperature.

Enter these unitless parameters as required.

- **Cementation exponent** m
- **Saturation exponent** n
- **Fluid saturation** S_L
- Enter a **Porosity** ε_p to set up the volume fraction of the fluid.

Porous Media


Note

This subfeature is available only when **Porous media** is selected as the material parameter in the parent feature node on any AC/DC interface (for example, the **Charge Conservation** or **Current Conservation** nodes). Then right-click the Charge Conservation or Current Conservation nodes to add this subnode.

Use the **Porous Media** subfeature to specify the material properties of a domain consisting of a porous medium using a mixture model. The **Porous Media** subfeature is available for all the AC/DC physics interfaces and, depending on the specific interface, can be used to provide a mixture model for the electric conductivity σ, the relative dielectric permittivity ε_r, or the relative magnetic permeability μ_r.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the porous media.

POROUS MEDIA

This section is always available and is used to define the mixture model for the domain.

Select the **Number of materials** (up to 5) to be included in the mixture model.

For each material (**Material 1**, **Material 2**, and so on) select either **Domain material**, to use the material specified for the domain, or one of the other materials specified in the **Materials** node. For each material, enter a **Volume fraction** θ_1, θ_2 and so on.

The Volume fractions specified for the materials should add to 1 in normal cases. Each subsequent volume fraction is automatically set to $\theta_2 = 1 - \theta_1$.


Note

The availability of the **Effective Electrical Conductivity**, **Effective Relative Permittivity**, and **Effective Relative Permeability** sections depend on the material properties used in the interface. Moreover, these sections are only active if the corresponding material property in the parent feature is set to **Porous media**.

EFFECTIVE ELECTRICAL CONDUCTIVITY, EFFECTIVE RELATIVE PERMITTIVITY, OR EFFECTIVE RELATIVE PERMEABILITY

Select the averaging method to use in the mixture model between the volume average of the material property, the volume average of its inverse, or the power law. For each material, specify either **From material**, to take the value from the corresponding material specified in the **Porous Media** section, or **User defined** to manually input a value.


See Also

- Effective Relative Permeability in Porous Media and Mixtures
- Effective Conductivity in Porous Media and Mixtures

External Current Density

The **External Current Density** node adds an externally generated current density $\mathbf{J}_e$, which appears in Ohm's law

$$\mathbf{J} = \sigma\mathbf{E} + \mathbf{J}_e$$

and in the equation that the interface defines.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define an external current density.


Note

For the **Electric Currents, Shell** interface, select boundaries instead of domains.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

EXTERNAL CURRENT DENSITY

Based on space dimension, enter the coordinates (**x**, **y**, and **z** for 3D models for example) of the **External current density** $\mathbf{J}_e$ (SI unit: $\mathrm{A/m^2}$).

Current Source

The **Current Source** node adds a distributed current source Q_j in the equation that the interface defines. Use this feature with caution as it may violate the current conservation law that is inherent in Maxwell-Ampère's law.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define a current source.

CURRENT SOURCE

Enter a value or expression for the **Current source** Q_j (SI unit: $\mathrm{A/m^3}$).

Initial Values

The **Initial Values** node adds an initial value for the electric potential that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver. If more than one set of initial values is required, right-click to add additional **Initial Values** features.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define an initial value.

INITIAL VALUES

Enter a value or expression for the initial value of the **Electric potential** V (SI unit: V). The default value is 0 V.

Boundary Current Source

The **Boundary Current Source** node adds a current source Q_j on the boundary.

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = Q_j$$

It is applicable to interior boundaries that represent either a source or a sink of current.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply a current source.


Note

For the **Electric Currents, Shell** interface, select edges (3D) or points (2D) instead of boundaries.

PAIR SELECTION

If **Boundary Current Source** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

BOUNDARY CURRENT SOURCE

Enter a value or expression for the **Boundary current source** Q_j (SI unit: A/m^2).

Normal Current Density

The **Normal Current Density** node is applicable to exterior boundaries that represent either a source or a sink of current. It provides a condition for specifying the normal current density as an inward or outward current flow:

$$-\mathbf{n} \cdot \mathbf{J} = J_n$$

Or, alternatively, as a current density $\mathbf{J}_0$:

$$\mathbf{n} \cdot \mathbf{J} = \mathbf{n} \cdot \mathbf{J}_0$$

The normal current density is positive when the current flows inward in the domain.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply a current flow as the boundary condition using the normal current density.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

NORMAL CURRENT DENSITY

Select an option from the **Type** list—**Inward current density** or **Current density**.

- If **Inward current density** is selected, enter a value or expression for the **Normal current density** J_n (SI unit: $\mathrm{A/m^2}$). Use a positive value for an inward current flow or a negative value for an outward current flow.
- If **Current density** is selected, enter values or expressions for the components of the **Current density** (SI unit: $\mathrm{A/m^2}$) in the $\mathbf{J}_0$ fields.

Distributed Impedance

The **Distributed Impedance** node adds a distributed impedance boundary condition to a model.

Use this boundary condition to model a thin sheet of a resistive material, connected to a reference potential V_{ref}.

The layer impedance can be specified either with the bulk material conductivity σ_s, the relative permittivity ε_r, and the layer thickness d_s, or directly with the surface resistance ρ_s and capacitance C_s. Assuming DC currents, the equation is:

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = \frac{\sigma_s}{d_s}(V - V_{\mathrm{ref}})$$

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = \frac{1}{\rho_s}(V - V_{\mathrm{ref}})$$

For the frequency domain and time dependent study types, this boundary condition is slightly more sophisticated and accounts also for capacitive coupling. The equations are:

$$\mathbf{n}\cdot(\mathbf{J}_1-\mathbf{J}_2) = \frac{(\sigma+j\omega\varepsilon_0\varepsilon_r)}{d_s}(V-V_{\text{ref}})$$

$$\mathbf{n}\cdot(\mathbf{J}_1-\mathbf{J}_2) = \left(\frac{1}{r_s}+j\omega C_s\right)(V-V_{\text{ref}})$$

$$\mathbf{n}\cdot(\mathbf{J}_1-\mathbf{J}_2) = \frac{1}{d_L}\left(\sigma(V-V_{\text{ref}})+\varepsilon_0\varepsilon_r\frac{\partial}{\partial t}(V-V_{\text{ref}})\right)$$

$$\mathbf{n}\cdot(\mathbf{J}_1-\mathbf{J}_2) = \left(\frac{1}{\rho_s}(V-V_{\text{ref}})+C_s\frac{\partial}{\partial t}(V-V_{\text{ref}})\right)$$

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply a distributed impedance.


Note

For the **Electric Currents, Shell** interface, select edges (3D) or points (2D) instead of boundaries.

DISTRIBUTED IMPEDANCE

Enter the **Reference potential** V_{ref} (SI unit: V).

Select a potentially complex valued **Layer specification**—**Thin layer** (the default) or **Surface impedance**.

- If **Thin layer** is selected, enter values or expressions for the:
  - **Surface thickness** d_s (SI unit: m). The default is $5\cdot10^{-3}$ m (5 mm).
  - **Electrical conductivity** σ (SI unit: S/m) and **Relative permittivity** ε_r. The defaults take values **From material**. Select **User defined** to enter different values or expressions.
- If **Surface impedance** is selected, enter values or expressions for the **Surface resistance** ρ_s (SI unit: Ωm^2) and for the **Surface capacitance** C_s (SI unit: F/m^2).

Electric Shielding

The **Electric Shielding** node provides an electric shielding boundary condition. Use this feature to model a thin layer of a highly conductive medium that shields the electric field. The sheet has the electrical conductivity σ_s and the surface thickness d.

The condition is represented by the following equation for interior boundaries and (setting $\mathbf{J}_2 = 0$) exterior boundaries assuming DC currents

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = -\nabla_t \cdot d(\sigma_s \nabla_t V)$$

For the frequency domain and time-dependent study types, also displacement currents are accounted for via the bulk relative permittivity of the sheet; ε_{rs} and the conservation laws change to:

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = -\nabla_t \cdot d((\sigma_s + j\omega\varepsilon_0\varepsilon_{rs})\nabla_t V)$$

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = -\nabla_t \cdot d\left(\sigma_s \nabla_t V + \varepsilon_0 \varepsilon_{rs} \frac{\partial}{\partial t} \nabla_t V\right)$$

Use this boundary condition when approximating a thin domain with a boundary to reduce the number of mesh elements.

Electric Shielding: Model Library path **ACDC_Module/Resistive_Devices/electric_shielding**

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply an electric shielding as the boundary condition.

MODEL INPUTS

Any model inputs (such as temperature for a temperature-dependent electrical conductivity) appear here.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

THICKNESS

Enter a value or expression for the **Surface thickness** d_s (SI unit: m).

PAIR SELECTION

If **Electric Shielding** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

ELECTRIC FIELD


Note

See **Electric Field** as described for the Charge Conservation node for the Electrostatics interface.

CONDUCTION CURRENT


Note

See **Conduction Current** as described for Current Conservation.

Electric Insulation

Electric Insulation is the default boundary condition and this feature adds electric insulation as the boundary condition:

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

This boundary condition means that no electric current flows into the boundary. At interior boundaries, it means that no current can flow through the boundary and that the electric potential is discontinuous across the boundary. It is also applicable at symmetric boundaries where the potential is known to be symmetric with respect to the boundary.


Tip

To add electric insulation to an interior boundary, add an **Electric Insulation** node in addition to the one that represents the default boundary condition. Electric insulation as the default boundary condition is not applicable to interior boundaries.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply electric insulation.


Note

For some interfaces, **All boundaries** are selected by default and cannot be changed. For the **Electric Currents, Shell** interface, select edges (3D) or points (2D) instead of boundaries.

PAIR SELECTION

If **Electric Insulation** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

Contact Impedance

Use the **Contact Impedance** node on interior boundaries to model a thin layer of resistive material. It can also be added as a pair.

$$\mathbf{n} \cdot \mathbf{J}_1 = \frac{\sigma}{d_s}(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \frac{\sigma}{d_s}(V_2 - V_1)$$

$$\mathbf{n} \cdot \mathbf{J}_1 = \frac{1}{\rho_s}(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \frac{1}{\rho_s}(V_2 - V_1)$$

The layer impedance can be specified either with the bulk material conductivity σ_s, the relative permittivity ε_r and the layer thickness d_s, or directly with the surface resistance ρ_s and capacitance C_s. The indices 1 and 2 refer to the two sides of the boundary.


Note

These parameters work the same as with Distributed Impedance.


Model

Thin-Film Resistance: Model Library path **COMSOL_Multiphysics/Electromagnetics/thin_film_resistance**

For the frequency domain and time-dependent study types, this boundary condition is slightly more sophisticated and accounts also for capacitive coupling. The corresponding equations are given below:

For the Frequency Domain study type:

$$\mathbf{n}\cdot\mathbf{J}_1 = \frac{(\sigma + j\omega\varepsilon_0\varepsilon_r)}{d_s}(V_1 - V_2)$$

$$\mathbf{n}\cdot\mathbf{J}_2 = \frac{(\sigma + j\omega\varepsilon_0\varepsilon_r)}{d_s}(V_2 - V_1)$$

$$\mathbf{n}\cdot\mathbf{J}_1 = \left(\frac{1}{\rho_s} + j\omega C_s\right)(V_1 - V_2)$$

$$\mathbf{n}\cdot\mathbf{J}_2 = \left(\frac{1}{\rho_s} + j\omega C_s\right)(V_2 - V_1)$$

For the Time-Dependent study type:

$$\mathbf{n}\cdot\mathbf{J}_1 = \frac{1}{d_s}\left(\sigma(V_1 - V_2) + \varepsilon_0\varepsilon_r\frac{\partial}{\partial t}(V_1 - V_2)\right)$$

$$\mathbf{n}\cdot\mathbf{J}_2 = \frac{1}{d_s}\left(\sigma(V_2 - V_1) + \varepsilon_0\varepsilon_r\frac{\partial}{\partial t}(V_2 - V_1)\right)$$

$$\mathbf{n}\cdot\mathbf{J}_1 = \left(\frac{1}{\rho_s}(V_1 - V_2) + C_s\frac{\partial}{\partial t}(V_1 - V_2)\right)$$

$$\mathbf{n}\cdot\mathbf{J}_2 = \left(\frac{1}{\rho_s}(V_2 - V_1) + C_s\frac{\partial}{\partial t}(V_2 - V_1)\right)$$

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply a contact resistance.

PAIR SELECTION

If **Contact Impedance** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

MATERIAL TYPE

Select a **Material type**—**Solid** (the default), **Non-solid**, or **From material**.

SURFACE IMPEDANCE

Select a potentially complex valued **Layer specification**—**Thin layer** (the default) or **Surface impedance**.

- If **Thin layer** is selected, enter values or expressions for the:
  - **Surface thickness** d_s (SI unit: m). The default is $5 \cdot 10^{-3}$ m (5 mm).
  - **Electrical conductivity** σ (SI unit: S/m) and **Relative permittivity** ε_r. The defaults take values **From material**. Select **User defined** to enter different values or expressions.
- If **Surface impedance** is selected, enter values or expressions for the **Surface resistance** ρ_s (SI unit: Ωm^2) and for the **Surface capacitance** C_s (SI unit: F/m^2).

Sector Symmetry

Select **Sector Symmetry** at interfaces between rotating objects where sector symmetry is used. It is only available for pairs.


Note

This feature assumes rotation around the origin.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries from an existing identity pair. This pair first has to be created.

PAIR SELECTION

When **Sector Symmetry** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

SECTOR SETTINGS

Enter the **Number of sectors (<50)** n_{sect}. The default is 2.

Select a **Type of periodicity**—**Continuity** or **Antiperiodicity**.

Based on space dimension, enter values or expressions in the table for the **Axis of rotation** $\mathbf{a}_{rot}$.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

See Also

In the *COMSOL Multiphysics User's Guide*:

- Identity and Contact Pairs
- Specifying Boundary Conditions for Identity Pairs

Line Current Source

2D

2D Axi

3D

The **Line Current Source** node adds a line source to edges in 3D models and to points in 2D and 2D axisymmetric models. The line source represents electric current per unit length.

EDGE OR POINT SELECTION

From the **Selection** list, choose the edges or points to add a current source.

Note

For the **Electric Currents, Shell** interface, this feature is only available for 3D models and on edges.

Caution

Beware that constraining the potential on edges or points usually yields a current outflow that is mesh dependent.

LINE CURRENT SOURCE

Enter a value or expression to apply a **Line current source** Q_j (SI unit: A/m). This source represents electric current per unit length.


See Also

Line Current Source (on Axis)

Line Current Source (on Axis)


2D Axi

The **Line Current Source (on Axis)** node adds a line source to boundaries in 2D axisymmetric models. The line source represents electric current per unit length.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to add a line current source (on axis).

LINE CURRENT SOURCE (ON AXIS)

Enter a value or expression to apply a **Line current source** Q_j (SI unit: A/m) to boundaries. This source represents electric current per unit length.


See Also

Line Current Source

Point Current Source


3D

The **Point Current Source** node adds a point source and represents an electric current flowing out of the point. Add point sources to 3D models from the **Points** menu.

POINT SELECTION

From the **Selection** list, choose the points to add a current source.


Caution

Beware that constraining the potential on points usually yields a current outflow that is mesh dependent.

POINT CURRENT SOURCE

Enter a value or expression to apply a **Point current source** Q_j (SI unit: A) to points. This source represents an electric current flowing out of the point.


See Also

- Line Current Source to apply it to points for 2D models.
- Point Current Source (on Axis) to apply to points for 2D axisymmetric models.

Point Current Source (on Axis)


2D Axi

The **Point Current Source (on Axis)** node adds a point source and represents an electric current flowing out of the point in 2D axisymmetric models.

POINT SELECTION

From the **Selection** list, choose the points to add a current source.


Caution

Beware that constraining the potential on points usually yields a current outflow that is mesh dependent.

POINT CURRENT SOURCE

Enter a value or expression to apply a **Point current source** Q_j (SI unit: A) to points. This source represents an electric current flowing out of the point.


See Also

- Point Current Source to apply this feature to points on 3D models.
- Line Current Source to apply it to points for 2D models.

Electric Point Dipole


2D


3D

The **Electric Point Dipole** node is available for 2D and 3D models.

The **Electric Point Dipole** represents the limiting case of zero separation distance between two equally strong point current sources and current sinks of opposing signs while maintaining the product between separation distance and source strength at a fixed value (*P*). The positive direction is from the current sink to the current source.

POINT SELECTION

From the **Selection** list, choose the points to add an electrostatic point dipole.

DIPOLE SPECIFICATION

Select a **Dipole specification**—**Magnitude and direction** or **Dipole moment**.

DIPOLE PARAMETERS

- If **Magnitude and direction** is selected under **Dipole Specification**, enter coordinates for the **Electric current dipole moment direction** $\mathbf{n}_p$ and the **Electric current dipole moment, magnitude** p (SI unit: A·m).
- If **Dipole moment** is selected under **Dipole Specification**, enter the components of the **Electric current dipole moment** $\mathbf{p}$ (SI unit: A·m).

See Also

Electric Point Dipole (on Axis)

Electric Point Dipole (on Axis)

2D Axi

The **Electric Point Dipole (on Axis)** node is available for 2D axisymmetric models.

The **Electric Point Dipole (on Axis)** represents the limiting case of zero separation distance between two equally strong point current sources and current sinks of opposing signs while maintaining the product between separation distance and source strength at a fixed value (P). The positive direction is from the current sink to the current source.

POINT SELECTION

From the **Selection** list, choose the points to add an electrostatic point dipole.

ELECTRIC POINT DIPOLE (ON AXIS)

Enter the **Electric current dipole moment in z direction** p_z (SI unit: m·A).

See Also

Electric Point Dipole

The Electric Currents, Shell Interface

The **Electric Currents, Shell** interface ( ), found under the **AC/DC** branch ( ) in the **Model Wizard**, adds the equations, boundary conditions, and current sources for modeling steady electric currents in thin current-conducting shells, solving for the electric potential. **Current Conservation** is the main feature, which adds the equation for the electric potential and provides a settings window for defining the electrical conductivity as well as the constitutive relation for the electric displacement field and its associated material properties such as the relative permittivity.

When this interface is added, these default nodes are also added to the **Model Builder**—**Current Conservation**, **Electric Insulation** (the default edge or point condition), and **Initial Values**. Right-click the **Electric Currents, Shell** node to add other features that implement, for example, edge or point conditions and current sources.


Note

Except where described below, the majority of the settings windows are the same as for the **Electrostatics** and **Electric Currents** interfaces as referenced. The only real difference are:

- For 3D models, boundaries are selected instead of domains, and edges instead of boundaries.
- For 2D and 2D axisymmetric models, boundaries are selected instead of domains, and points instead of edges and boundaries.


See Also

For a more extensive introduction to the physics and equations implemented by this interface, see the Theory for the Electric Currents, Shell Interface.

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `ecs`.

BOUNDARY SELECTION

Select the boundaries (shells) where you want to define the electric potential and the equations that describe the potential field for conductive media. The default setting is to include all boundaries in the model.

OUT-OF-PLANE THICKNESS


2D

Enter a value or expression for the **Thickness** d. The default value is 1 unit length.

Use the Change Thickness (Out-of-Plane) node to define specific boundaries or points instead of a global setting for the thickness.

SHELL THICKNESS

Enter a value or expression for the **Shell thickness** d_s (SI unit: m). The default value is 1 cm.


Note

When modeling with shells, the **Shell Thickness** section defines a parameter that enters the equations everywhere. It is available in all dimensions.

This is unrelated to the **Change Thickness (Out-of-Plane)** node, which is only available in 2D and represents the out-of-plane length of the shell, which is being modeled as a cross-section.

When modeling in 2D, this means that on the interface settings window, there are two *sections*—the **Shell Thickness** and **Out-of-Plane Thickness**—available, as well as two *feature nodes*—**Change Shell Thickness** and **Change Thickness (Out-of-Plane)**.

TERMINAL SWEEP SETTINGS

Enter a **Reference impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω.

Select the **Activate terminal sweep** check box to switch on the sweep and invoke a parametric sweep over the terminals. Enter a **Sweep parameter name** to assign a specific name to the variable that controls the terminal number solved for during the sweep. The default is `PortName`. The generated lumped parameters are in the form of capacitance matrix elements. The terminal settings must consistently be of either fixed voltage or fixed charge type.

The lumped parameters are subject to **Touchstone file export**. Enter a file path or **Browse** for a file. Select an **Output format** for the Touchstone export—**Magnitude angle**, **Magnitude (dB) angle**, or **Real imaginary**. Select a **Parameter to export**—**Z** (the default), **Y**, or **S**.


See Also

Lumped Parameters

DEPENDENT VARIABLES

The dependent variable (field variable) is for the **Electric potential** *V*. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. Select an element order for the **Electric Potential**—**Linear**, **Quadratic** (the default), **Cubic**, **Quartic**, or (in 2D only) **Quintic**.Specify the **Value type when using splitting of complex variables**—**Real** or **Complex** (the default).


See Also

- The Model Builder Show and Hide Physics Options
- Boundary, Edge, Point, and Pair Conditions for the Electric Currents, Shell Interface

Boundary, Edge, Point, and Pair Conditions for the Electric Currents, Shell Interface

The Electric Currents, Shell Interface has the following boundary, edge, point, and pair conditions available as indicated.

About the Edge and Point Conditions

The conditions in Table 4-1 are available at interfaces between different media and interior edges (in 3D models) and point conditions (in 2D and 2D axisymmetric models) in continuity; that is,

$$\mathbf{n}_2 \cdot (\mathbf{J}_1 - \mathbf{J}_2) = 0$$

which is the natural edge/point condition.

Available Features

These features are described in this section:

- Change Shell Thickness
- Current Conservation
- Current Source
- Electric Shielding
- Initial Values
- Normal Current Density

These features are available for this interface and described for the **Electric Currents** and **Electrostatics** interfaces (listed in alphabetical order):

- Boundary Current Source
- Change Thickness (Out-of-Plane)
- Current Source
- Distributed Impedance
- Electric Insulation
- Electric Point Dipole
- Electric Potential
- External Current Density
- Ground
- Initial Values
- Line Current Source
- Point Current Source
- Terminal


Tip

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

Current Conservation

The **Current Conservation** node adds the continuity equation for the electrical potential and provides an interface for defining the electric conductivity as well as the constitutive relation and the relative permittivity for the displacement current.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define the electric potential and the continuity equation that describes the potential field.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty. If a linear temperature relation is added for the conductivity, then the source for the temperature T can be defined. From the **Temperature** list, select an existing temperature variable (from another physics interface) if available, or select **User defined** to define a value or expression for the temperature (SI unit: K) in the field that appears underneath the list.

MATERIAL TYPE

Select a **Material type**—**Solid** (the default), **Non-solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

ELECTRIC FIELD


Note

See **Electric Field** as described for the Charge Conservation node for the **Electrostatics** interface.

CONDUCTION CURRENT

By default, the **Electrical conductivity** σ (SI unit: S/m) for the media is defined **From material**. Or select **User defined**, **Linearized resistivity**, **Porous media**, or **Archie's law**.

User Defined

If **User defined** is selected, select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** depending on the characteristics of the electrical conductivity, and then enter values or expressions in the field or matrix.


Note

If another type of temperature dependence is used other than a linear temperature relation, enter any expression for the conductivity as a function of temperature.

Linearized Resistivity

Select **Linearized resistivity** for a temperature-dependent conductivity (this occurs in, for example, Joule heating, and is also called resistive heating). The equation describing the conductivity:

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

where ρ_0 is the resistivity at the reference temperature T_0, and α is the temperature coefficient of resistance, which describes how the resistivity varies with temperature.

The default **Reference temperature** T_{ref} (SI unit: K), **Resistivity temperature coefficient** α (SI unit: $1/\text{K}$), and **Reference resistivity** ρ_0 (SI unit: Ωm) are taken **From material**, which means that the values are taken from the domain material.


Note

Only certain material models (see Materials) support the **Linearized resistivity**.

To specify other values for any of these properties, select **User defined** from the list and then enter a value or expression. T is the current temperature, which can be a value that is specified as a model input or the temperature from a heat transfer interface. The definition of the temperature field appears in the **Model Inputs** section.

Porous Media

When **Porous media** is selected, right-click to add a Porous Media subnode.

Archie's Law

When **Archie's law** is selected, right-click to add an Archie's Law subnode.

Initial Values

Initial Values adds an initial value for the electric potential that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver. Right-click to add additional **Initial Values** features.

BOUNDARY SELECTION

Select the boundaries where you want to define an initial value.

INITIAL VALUES

Enter a value or expression for the initial value of the **Electric potential** V (SI unit: V). The default is 0.

Current Source

The **Current Source** node adds a distributed current source Q_j in the equation that the interface defines. Use this feature with caution as it may violate the current conservation law that is inherent in Maxwell-Ampère's law.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define a current source.

ELECTRODE CURRENT SOURCE

Enter a value or expression for the **Current source** Q_j (SI unit: $\mathrm{A/m^3}$).

Change Shell Thickness

Add a **Change Shell Thickness** node to specify a different shell thickness for a subset of the boundaries or edges where the **Electric Currents, Shell** interface is defined.

BOUNDARY OR EDGE SELECTION

Select the boundaries or edges where you want to specify a shell thickness that differs from that for the parent Electric Currents, Shell interface.

CHANGE SHELL THICKNESS

Enter a **Shell thickness** d_s (SI unit: m). The default is 1 cm.

Normal Current Density

The **Normal Current Density** feature represents a current density flowing normally from the domain into the shell boundary. This condition is different from the **Current Source** feature since it represents a net current density flowing from the adjacent domain into the shell, and can be used to model, for example, a boundary acting as an electrode.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to apply a current flow as the boundary condition using the normal current density.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

NORMAL CURRENT DENSITY

Enter a value or expression for the **Normal current density** J_n (unitless).

Electric Shielding


3D

This feature is available for 3D models only.

The **Electric Shielding** feature can be used to models a geometrically thin section of shell made of a highly conductive medium. The Layer Thickness parameter `d_l` specifies the thickness in the direction tangential to the shell, while the thickness in the direction normal to the shell is taken from the physics interface or, if present, from a **Change Shell Thickness** node.

The layer has an electrical conductivity of σ and a relative permittivity of ε_r.

EDGE SELECTION

From the **Selection** list, choose the edges to apply an electric shielding as the boundary condition.

PAIR SELECTION

If **Electric Shielding** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

ELECTRIC SHIELDING

The default **Relative permittivity** ε_r (unitless) and **Electrical conductivity** σ (SI unit: S/m) take values **From material**. Select **User defined** to enter different values or expressions.

Enter a value or expression for the **Layer thickness** d_1 (SI unit: m). The default is 1 cm.

Theory of Electric Fields

COMSOL Multiphysics includes physics interfaces for the modeling of static electric fields and currents. Deciding what specific physics interface and study type to select for a particular modeling situation requires a basic understanding of the charge dynamics in conductors. This section is a brief introduction to Charge Relaxation Theory.


Tip

Physics interfaces for the modeling of dynamic, quasi-static (that is, without including wave propagation effects) electric fields and currents are available with the AC/DC Module and MEMS Module.

Charge Relaxation Theory

The different physics interfaces involving only the scalar electric potential can be interpreted in terms of the charge relaxation process. The fundamental equations involved are *Ohm's law* ($\mathbf{J} = \sigma\mathbf{E}$) the *equation of continuity*

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \mathbf{J} = 0$$

and *Gauss' law*

$$\nabla \cdot (\varepsilon \mathbf{E}) = \rho$$

By combining these, one can deduce the following differential equation for the space charge density in a homogeneous medium

$$\frac{\partial \rho}{\partial t} + \frac{\sigma}{\varepsilon}\rho = 0$$

This equation has the solution

$$\rho(t) = \rho_0 e^{-t/\tau}$$

where

$$\tau = \frac{\varepsilon}{\sigma}$$

is called the charge relaxation time. For a good conductor like copper, τ is of the order of 10^{-19} s whereas for a good insulator like silica glass, it is of the order of 10^{3} s. For a pure insulator, it becomes infinite.

When modeling real world devices, there is not only the intrinsic time scale of charge relaxation time but also an external time scale t at which a device is energized or the observation time. It is the relation between the external time scale and the charge relaxation time that determines what physics interface and study type to use. The results are summarized in Table 4-3 below,

TABLE 4-3: SUITABLE PHYSICS INTERFACE AND STUDY TYPE FOR DIFFERENT TIME-SCALE REGIMES.

CASE	PHYSICS INTERFACE	STUDY TYPE
$\tau >> t$	Electrostatics	Stationary
$\tau << t$	Electric Currents	Stationary
$\tau \sim t$	Electric Currents	Time Dependent or Frequency Domain (in AC/DC Module or MEMS Module)

FIRST CASE: $\tau >> T$

If the external time scale is short compared to the charge relaxation time, the charges do not have time to redistribute to any significant degree.Thus the charge distribution can be considered as given model input and the best approach is to solve the *Electrostatics* formulation using the electric potential *V*.

By combining the definition of the potential with Gauss' law, you can derive the classical Poisson's equation. Under static conditions, the electric potential *V* is defined by the equivalence $\mathbf{E} = -\nabla V$. Using this together with the constitutive relation $\mathbf{D} = \varepsilon_0 \mathbf{E} + \mathbf{P}$ between $\mathbf{D}$ and $\mathbf{E}$, you can rewrite Gauss' law as a variant of Poisson's equation

$$-\nabla \cdot (\varepsilon_0 \nabla V - \mathbf{P}) = \rho$$

This equation is used in the Electrostatics interface. It is worth noting that Gauss' law does not require the charge distribution to be static. Thus, provided dynamics are slow enough that induced electric fields can be neglected and hence a scalar electric potential is justified, the formulation can be used also in the Time Dependent study

type. That typically involves either prescribing the charge dynamics or coupling a separate formulation for this.

Such separate charge transport formulations can be found in the Plasma Module and the Chemical Reaction Engineering Module.

SECOND CASE: τ <<T

If the external time scale is long compared to the charge relaxation time, the stationary solution to the equation of continuity has been reached. In a stationary coordinate system, a slightly more general form than above of *Ohm's law* states that

$$\mathbf{J} = \sigma\mathbf{E} + \mathbf{J}^{\mathrm{e}}$$

where $\mathbf{J}^{\mathrm{e}}$ is an externally generated current density. The static form of the equation of continuity then reads

$$\nabla \cdot \mathbf{J} = -\nabla \cdot (\sigma\nabla V - \mathbf{J}^{\mathrm{e}}) = 0$$

To handle current sources the equation can be generalized to

$$-\nabla \cdot (\sigma\nabla V - \mathbf{J}^{\mathrm{e}}) = Q_j$$

This equation is used in the static study type for the Electric Currents interface.

GENERAL CASE: CHARGE DYNAMICS

If the charge relaxation time is comparable to the external time scale, the time dependent or frequency domain study types for the Electric Currents interface must be used.

Combining the time-harmonic equation of continuity

$$\nabla \cdot \mathbf{J} = \nabla \cdot (\sigma\mathbf{E} + \mathbf{J}^{\mathrm{e}}) = -j\omega\rho$$

with the equation ($\nabla \cdot \mathbf{D} = \rho$) yields the following equation for the frequency domain study type:

$$-\nabla \cdot ((\sigma + j\omega\varepsilon_0)\nabla V - (\mathbf{J}^{\mathrm{e}} + j\omega\mathbf{P})) = 0$$

For the time dependent study type, use the transient equation of continuity

$$\nabla \cdot \mathbf{J} = \nabla \cdot (\sigma \mathbf{E} + \mathbf{J}^{\mathrm{e}}) = -\frac{\partial \rho}{\partial t}$$

and the resulting equation becomes

$$-\nabla \cdot \frac{\partial}{\partial t}(\varepsilon_0 \nabla V + \mathbf{P}) - \nabla \cdot (\sigma \nabla V - \mathbf{J}^{\mathrm{e}}) = 0$$

These dynamic formulations are valid as long as induced electric fields can be ignored and hence the electric field is essentially curl free. This condition is fulfilled provided that skin effect and wave propagation effects can be ignored. The skin depth must be much larger than the geometrical dimensions of the modeled device and so must the wavelength. Note also that these formulations can be used to model dielectric regions of capacitive/resistive devices even though the interior of electrodes may not meet the large skin depth condition. In that case, the electrodes must only be represented as boundary conditions (fixed or floating potential). The interior, metallic domains are not included in the analysis. Obviously, this is only a valid approach for devices where metallic electrodes do not entirely bypass (short circuit) the capacitive/resistive layers. If metallic electrodes short circuit the capacitive/resistive layers, the time evolution of the current is determined by inductive and resistive effects with very little influence from the capacitive layers. Then the Magnetic Fields interface is the appropriate modeling tool.

Theory for the Electrostatics Interface

The Electrostatics Interface is available for 3D, 2D in-plane, and 2D axisymmetric models. Applications with Electrostatics Equations include high-voltage apparatus, electronic devices, and capacitors. The term "statics" is not to be interpreted literally—it is the observation time or time scale at which the applied excitation changes is short compared to the charge relaxation time and that the electromagnetic wavelength and skin depth are very large compared to the size of the domain of interest.

If you do not know whether to use the Electric Currents or the Electrostatics interface, which both solve for the scalar electric potential V, consider using an explicit charge transport model. See Charge Relaxation Theory.

Electrostatics Equations

Under static conditions the electric potential, V, is defined by the relationship

$$\mathbf{E} = -\nabla V$$

Combining this equation with the constitutive relationship $\mathbf{D} = \varepsilon_0\mathbf{E} + \mathbf{P}$ between the electric displacement $\mathbf{D}$ and the electric field $\mathbf{E}$, it is possible to represent Gauss' law as the following equation:

$$-\nabla \cdot (\varepsilon_0 \nabla V - \mathbf{P}) = \rho$$

In this equation, the physical constant, ε_0 (SI unit: F/m) is the permittivity of vacuum, $\mathbf{P}$ (SI unit: C/m^2) is the electric polarization vector, and ρ (SI unit: C/m^3) is a space charge density. This equation describes the electrostatic field in dielectric materials.

For in-plane 2D modeling, the Electrostatics interface assumes a symmetry where the electric potential varies only in the x and y directions and is constant in the z direction. This implies that the electric field, $\mathbf{E}$, is tangential to the xy-plane. With this symmetry, the same equation is solved as in the 3D case. The interface solves the following equation where d is the thickness in the z direction:

$$-\nabla \cdot d(\varepsilon_0 \nabla V - \mathbf{P}) = \rho$$

The axisymmetric version of the interface considers the situation where the fields and geometry are axially symmetric. In this case the electric potential is constant in the φ direction, which implies that the electric field is tangential to the *rz*-plane.

Theory for the Electric Currents Interface

The Electric Currents Interface solves a current conservation problem for the scalar electric potential V and is available for 3D, 2D in-plane, and 2D axisymmetric models. Electrolysis and the computation of resistances of grounding plates are examples that involve conductive media with electrical conductivities and electric currents. If you are uncertain whether to use the Electric Currents interface or the Electrostatics interface, which both solve for the scalar electric potential V, refer to the section on Charge Relaxation Theory.

In this section:

- Electric Currents Equations in Steady State
- Effective Conductivity in Porous Media and Mixtures
- Effective Relative Permeability in Porous Media and Mixtures
- Archie's Law Theory
- Dynamic Electric Currents Equations
- Reference for the Electric Currents Interface

Electric Currents Equations in Steady State

When handling stationary electric currents in conductive media you must consider the stationary equation of continuity. In a stationary coordinate system, the point form of *Ohm's law* states that

$$\mathbf{J} = \sigma\mathbf{E} + \mathbf{J}_e$$

where σ is the electrical conductivity (SI unit: S/m), and $\mathbf{J}_e$ is an *externally generated current density* (SI unit: A/m^2). The static form of the equation of continuity then states

$$\nabla \cdot \mathbf{J} = -\nabla \cdot (\sigma \nabla V - \mathbf{J}_e) = 0$$

To handle *current sources*, you can generalize the equation to

$$-\nabla \cdot (\sigma \nabla V - \mathbf{J}_e) = Q_j$$

In planar 2D the Electric Currents interface assumes that the model has a symmetry where the electric potential varies only in the x and y directions and is constant in the z direction. This implies that the electric field, $\mathbf{E}$, is tangential to the xy-plane. The Electric Currents interface then solves the following equation where d is the thickness in the z direction:

$$-\nabla \cdot d(\sigma \nabla V - \mathbf{J}_e) = dQ_j \qquad (4\text{-}1)$$

In 2D axisymmetry, the Electric Currents interface considers the situation where the fields and geometry are axially symmetric. In this case the electric potential is constant in the φ direction, which implies that the electric field is tangential to the rz-plane.

Effective Conductivity in Porous Media and Mixtures

When handling electric currents in porous media or mixtures of solids with different electric properties, you must consider different ways for obtaining the **Effective conductivity** of the mixture.

There are several possible approaches to do this, starting from the values defined by the user, composed by a volume fraction θ_1 of material 1, and a volume fraction $\theta_2 = 1 - \theta_1$ of material 2.

The effective conductivity σ is then given as input for the electric current conservation specified in Equation 4-1, in the same way of modeling an effective (single phase) material.

VOLUME AVERAGE, CONDUCTIVITY

If the electric conductivities of the two materials are not so different from each other, a simple form of averaging can be used, such as a volume average:

$$\sigma = \theta_1\sigma_1 + \theta_2\sigma_2$$

here σ_1 is the conductivity of the material 1 and σ_2 is that of material 2. This is equivalent to a "parallel" system of resistivities.


Note

If the conductivities are defined by second order tensors (such as for anisotropic materials), the volume average is applied element by element.

VOLUME AVERAGE, RESISTIVITY

A similar expression for the effective conductivity can be used, which mimics a "series" connection of resistivities. Equivalently, the effective conductivity is obtained from

$$\frac{1}{\sigma} = \frac{\theta_1}{\sigma_1} + \frac{\theta_2}{\sigma_2}$$


Note

If the conductivities are defined by second order tensors, the inverse of the tensors are used.

POWER LAW

A power law gives the following expression for the equivalent conductivity:

$$\sigma = \sigma_1^{\theta_1}\sigma_2^{\theta_2}$$


Note

The effective conductivity calculated by Volume Average, Conductivity is the upper bound, the effective conductivity calculated by Volume Average, Resistivity is the lower bound, and the Power Law average is somewhere in between these two.

Effective Relative Permeability in Porous Media and Mixtures

When handling electric currents in porous media or mixtures of solids with different electric properties, you must consider different ways for obtaining the *effective relative permeability* of the mixture.

There are several possible approaches to do this, starting from the values defined by the user, composed by a volume fraction θ_1 of material 1, and a volume fraction $\theta_2 = 1 - \theta_1$ of material 2.

The effective relative permeability ε_r is then given as input for the electric current conservation specified in Equation 4-2, in the same way of modeling an effective (single phase) material.

VOLUME AVERAGE, PERMEABILITY

If the relative permeability of the two materials are not so different from each other, the effective relative permeability μ_r is calculated by simple volume average:

$$\mu_r = \theta_1\mu_1 + \theta_2\mu_2$$

here μ_1 is the relative permeability of the material 1, and μ_2 is that of material 2.


Note

If the permeability is defined by second order tensors (such as for anisotropic materials), the volume average is applied element by element.

VOLUME AVERAGE, RECIPROCAL PERMEABILITY

A similar expression for the effective permeability can be used, which mimics a "series" connection of resistivities. Equivalently, the effective reciprocal permeability is obtained from

$$\frac{1}{\mu_r} = \frac{\theta_1}{\mu_1} + \frac{\theta_2}{\mu_2}$$


Note

If the permeability is defined by second order tensors, the inverse of the tensors are used.

POWER LAW

A power law gives the following expression for the equivalent permeability:

$$\mu_r = \mu_1^{\theta_1}\mu_2^{\theta_2}$$


Note

The effective permeability calculated by Volume Average, Permeability is the upper bound, the effective permeability calculated by Volume Average, Reciprocal Permeability is the lower bound, and the Power Law average gives a value somewhere in between these two.

Archie's Law Theory

The electrical conductivity of the materials composing saturated rocks and soils can vary over many orders of magnitude. For instance, in the petroleum reservoirs, normal sea water (or brine) has a typical conductivity of around 3 S/m, whereas hydrocarbons are typically much more resistive and have conductivities in the range 0.1 – 0.01 S/m.

The porous rocks and sediments may have even lower conductivities. In variably saturated soils, the conductivity of air is roughly ten orders of magnitude lower that the ground water. A simple volume average (of either conductivity or resistivity) in rocks or soils might give different results compared to experimental data.

Since most crustal rocks, sedimentary rocks, and soils are formed by non-conducting materials, Archie (Ref. 1) assumed that electric current are mainly caused by ion fluxes trough the pore network. Originally, Archie's law is an empirical law for the effective conductivity of a fully-saturated rock or soil, but it can be extended to variably saturated porous media.

Archie's law relates the effective conductivity to the fluid conductivity σ_L, fluid saturation s_L and porosity ε_p:

$$\sigma = s_L^n \varepsilon_p^m \sigma_L$$

here, m is the cementation exponent, a parameters that describes the connectivity of the pores. The cementation exponent normally varies between 1.3 and 2.5 for most sedimentary rocks, and it is close to 2 for sandstones. The lower limit $m = 1$ represents a volume average of the conductivities of a fully saturated, insulating (zero conductivity) porous matrix, and a conducting fluid. The saturation coefficient n is normally close to 2.

The ratio $F = \sigma_L/\sigma$ is called the *formation factor*.

Archie's Law does not take care of the relative permittivity of either fluids or solids, so the effective relative permittivity of the porous medium is normally consider as $\varepsilon_r = 1$.

Dynamic Electric Currents Equations

In the frequency domain and time dependent study types dynamic formulations accounting for both conduction currents and displacement currents are used.

Combining the time-harmonic equation of continuity

$$\nabla \cdot \mathbf{J} = \nabla \cdot (\sigma \mathbf{E} + \mathbf{J}^e) = -j\omega\rho$$

with the equation

$$\nabla \cdot \mathbf{D} = \rho$$

and generalized to handle *current sources* yields the following equation:

$$-\nabla \cdot ((\sigma + j\omega\varepsilon_0)\nabla V - (\mathbf{J}^e + j\omega\mathbf{P})) = Q_j$$

For the transient case, using the transient equation of continuity

$$\nabla \cdot \mathbf{J} = \nabla \cdot (\sigma\mathbf{E} + \mathbf{J}^e) = -\frac{\partial \rho}{\partial t}$$

and generalized to handle *current sources* the resulting equation becomes

$$-\nabla \cdot \frac{\partial}{\partial t}(\varepsilon_0 \nabla V + \mathbf{P}) - \nabla \cdot (\sigma \nabla V - \mathbf{J}^e) = Q_j \tag{4-2}$$

In planar 2D the dynamic formulations also involves the thickness d in the z direction:

$$-\nabla \cdot d((\sigma + j\omega\varepsilon_0)\nabla V - (\mathbf{J}^e + j\omega\mathbf{P})) = dQ_j$$

$$-\nabla \cdot d\frac{\partial}{\partial t}((\varepsilon_0 \nabla V + \mathbf{P}) - \nabla \cdot d(\sigma \nabla V - \mathbf{J}^e)) = dQ_j$$

Reference for the Electric Currents Interface

1. G.E. Archie, “The Electric Resistivity as an Aid in Determining Some Reservoir Characteristics,” *Trans. Am. Inst. Metal. Eng.* 146, 54–62, 1942.

Theory for the Electric Currents, Shell Interface

The Electric Currents, Shell Interface in 3D to model thin shells of conductive media. This physics interface is similar to the 2D Electric Currents interface, solving for the electric potential on 2D surfaces in a 3D geometry. The difference is that the shell does not have to be flat as they obviously are when using the 2D Electric Currents interface. The Electric Currents, Shell interface is also available on boundaries in 2D geometries.

In this section:

- Electric Currents, Shell Equations in Steady State
- Dynamic Electric Currents Equations

Electric Currents, Shell Equations in Steady State

In the static study type, the interface solves the following equation where d is the thickness (SI unit: m) of the shell:

$$-\nabla_t \cdot d(\sigma \nabla_t V - \mathbf{J}^e) = dQ_j$$

σ is the electrical conductivity (SI unit: S/m), $\mathbf{J}_e$ is an *externally generated current density* (SI unit: A/m^2), and Q_j is an external current source (SI unit: A/m^3). The operator ∇_t represents the tangential derivative along the shell.

Dynamic Electric Currents Equations

In the frequency domain and time dependent study types dynamic formulations accounting for both conduction currents and displacement currents are used:

$$-\nabla_t \cdot d((\sigma + j\omega\varepsilon_0)\nabla_t V - (\mathbf{J}^e + j\omega\mathbf{P})) = dQ_j$$

For the transient case, the resulting equation becomes

$$-\nabla_t \cdot d\frac{\partial}{\partial t}((\varepsilon_0 \nabla_t V + \mathbf{P}) - \nabla_t \cdot d(\sigma \nabla_t V - \mathbf{J}^e)) = dQ_j$$

5

The Magnetic Field Interfaces

This chapter summarizes the functionality of the magnetic field interfaces found under the **AC/DC** branch ( ) in the **Model Wizard**.

In this chapter:

The Magnetic Fields Interface

Note

The AC/DC Module enhances the Magnetic Fields interface included with the basic COMSOL Multiphysics license. These features are described in Magnetic Fields Interface Advanced Features.

The **Magnetic Fields** interface ( ), found under the **AC/DC** branch ( ) in the **Model Wizard**, has the equations, boundary conditions, and currents for modeling magnetic fields, solving for the magnetic vector potential. The main feature is the **Ampère's Law** feature, which adds the equation for the magnetic vector potential and provides an interface for defining the constitutive relations and its associated properties such as the relative permeability.

When this interface is added, these default nodes are also added to the **Model Builder**—**Magnetic Fields**, **Ampère's Law**, **Magnetic Insulation** (the default boundary condition), and **Initial Values**. Right-click the **Magnetic Fields** node to add other features that implement, for example, boundary conditions and external currents.

Model

- Quadrupole Lens: Model Library path **COMSOL_Multiphysics/Electromagnetics/quadrupole**
- Eddy Currents: Model Library path **ACDC_Module/Inductive_Devices_and_Coils/eddy_currents**

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `mf`.

DOMAIN SELECTION

The default setting is to include **All domains** in the model to define the magnetic vector potential and the equations that describe the potential field for magnetic fields. To choose specific domains, select **Manual** from the **Selection** list.

BACKGROUND FIELD

Select an option from the **Solve for** list—**Full field** (the default) or **Reduced field**. If **Reduced field** is selected, specify a **Background magnetic vector potential** $\mathbf{A}_b$ (SI unit: Wb/m). The total field used in the physics and equations are given by the sum of the reduced and background fields.

COMPONENTS


2D

2D Axi

This section is only available in 2D and 2D axially symmetric models.


Note

The current vector has the same direction as the magnetic vector potential, so this setting also controls the direction in which applied and induced currents can flow in the model. The default option is to solve for the out-of-plane component only for 2D and 2D axisymmetric models.

Select **Components**—**Out-of-plane vector potential** (the default), **In-plane vector potential**, or **Three-component vector potential** for the magnetic vector potential. From the practical viewpoint this choice is equivalent to deciding in what directions the electric current is allowed to flow (out-of-plane currents, in-plane currents or currents flowing in all three coordinate directions) and affects other settings in the model, for example, the **Port Properties>Type of port** section for the Lumped Port node.


2D

2D Axi

For 2D and 2D axisymmetric models and when **In-plane vector potential** or **Three-component vector potential** is selected, a Gauge Fixing for A-field subnode can be added to the Ampère's Law node.

THICKNESS

2D

Enter a value or expression for the global **Out-of-plane thickness** d (SI unit: m). The default value of 1 m is typically not representative for a thin domain. Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D models.

Use the Change Thickness (Out-of-Plane) node (described for the **Electrostatics** interface) to define specific geometric entities (for example, domains) instead of a global setting for the thickness.

SWEEP SETTINGS

Enter a **Reference impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω.

Select the **Activate port sweep** check box to switch on the sweep and invoke a parametric sweep over the ports. Enter a **Sweep parameter name** to assign a specific name to the variable that controls the port number solved for during the sweep. The generated lumped parameters are in the form of capacitance matrix elements. The port settings must consistently be of either fixed voltage or fixed charge type. The default is `PortName`.

The lumped parameters are subject to **Touchstone file export**. Enter a file path or **Browse** for a file. Select an **Output format** for the Touchstone export—**Magnitude angle**, **Magnitude (dB) angle**, or **Real imaginary**.

See Also

Lumped Parameters

DEPENDENT VARIABLES

The dependent variables (field variables) are for the **Magnetic vector potential** A and its components. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. Select a **Magnetic vector potential**—**Quadratic** (the default), **Linear**, **Cubic**, or **Quartic**. Specify the **Value type when using splitting of complex variables**—**Real** or **Complex** (the default).

See Also

- The Model Builder Show and Hide Physics Options
- Domain, Boundary, Point, and Pair Conditions for the Magnetic Fields Interface
- Theory for the Magnetic Fields Interface

Domain, Boundary, Point, and Pair Conditions for the Magnetic Fields Interface

The Magnetic Fields Interface has these domain, boundary, point, and pair features available. Features are listed in alphabetical order.

About the Boundary Conditions

With no surface currents present the interface conditions

$$\mathbf{n}_2 \times (\mathbf{A}_1 - \mathbf{A}_2) = \mathbf{0}$$
$$\mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{0}$$

need to be fulfilled. Because $\mathbf{A}$ is being solved for, the tangential component of the magnetic potential is always continuous, and thus the first condition is automatically fulfilled. The second condition is equivalent to the natural boundary condition and is hence also fulfilled unless surface currents are explicitly introduced.

Available Features

These features are available for this interface and listed in alphabetical order. Also see Table 5-1 for a list of interior and exterior boundary conditions.

- Ampère's Law
- Boundary Feed
- Change Thickness (Out-of-Plane) (described for the **Electrostatics** interface)
- Coil Group Domain
- Edge Current
- Electric Point Dipole (described for the **Electric Currents** interface)

- Electric Point Dipole (on Axis) (described for the **Electric Currents** interface)
- External Current Density
- External Magnetic Vector Potential
- Force Calculation (described for the **Electrostatics** interface)
- Gap Feed
- Impedance Boundary Condition
- Initial Values
- Line Current (Out-of-Plane)
- Lumped Port
- Magnetic Field
- Magnetic Insulation (the default boundary condition)
- Magnetic Point Dipole
- Magnetic Point Dipole (on Axis)
- Magnetic Potential
- Magnetic Shielding
- Multi-Turn Coil Domain
- Perfect Magnetic Conductor
- Reference Edge
- Reversed Current Direction
- Sector Symmetry
- Single-Turn Coil Domain
- Surface Current
- Thin Low Permeability Gap
- Transition Boundary Condition
- Velocity (Lorentz Term)


Tip

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

Table 5-1 lists the interior and exterior boundaries available with this interface.

TABLE 5-1: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS FOR THE MAGNETIC FIELDS INTERFACE

FEATURE	INTERIOR	EXTERIOR
Change Thickness (Out-of-Plane)	x	x
External Magnetic Vector Potential	x	x
Impedance Boundary Condition		x
Lumped Port	x	x
Magnetic Field	x	x
Magnetic Insulation	x	x
Magnetic Potential	x	x
Magnetic Shielding	x	x
Perfect Magnetic Conductor	x	x
Periodic Condition		x
Surface Current	x	x
Thin Low Permeability Gap	x	
Transition Boundary Condition	x	

2D Axi

For 2D axisymmetric models, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an **Axial Symmetry** node to the model that is valid on the axial symmetry boundaries only.

See Also

About Infinite Element Domains and Perfectly Matched Layers in the *COMSOL Multiphysics User's Guide*

Ampère's Law

The **Ampère's Law** feature adds Ampère's law for the magnetic field and provides an interface for defining the constitutive relation and its associated properties such as the relative permeability as well as electric properties.

 *3D*	For 3D models, right-click the **Ampère's Law** node to add a Gauge Fixing for A-field feature.
 *2D*  *2D Axi*	For 2D and 2D axisymmetric models, and when **In-plane vector potential** or **Three-component vector potential** is selected from the **Components** section on The Magnetic Fields Interface settings window, right-click the **Ampère's Law** node to add a Gauge Fixing for A-field feature. For some interfaces this feature is added by default.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the magnetic vector potential and the equation based on Ampère's law that defines the potential.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty. If a linear temperature relation is added for the conductivity, then define the source for the temperature T. From the **Temperature** list, select an existing temperature variable (from another physics interface) if available, or select **User defined** to define a value or expression for the temperature (SI unit: K) in the field that appears underneath the list.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

CONDUCTION CURRENT

By default, the **Electrical conductivity** σ (SI unit: S/m) for the media is defined **From material**. Also select **User defined**, **Porous media**, **Archie's law**, or **Linearized resistivity** from the list. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the electrical conductivity, and then enter values or expressions in the field or matrix.

Linearized Resistivity

If **Linearized resistivity** is selected, it defines the electric resistivity (and conductivity) as a linear function of temperature and this equation describes the conductivity:

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

where ρ_0 is the resistivity at the reference temperature T_0. α is the temperature coefficient of resistance, which describes how the resistivity varies with temperature.

- The default **Reference temperature** T_{ref} (SI unit: K), **Resistivity temperature coefficient** α (SI unit: 1/K), and **Reference resistivity** ρ_0 (SI unit: Ω·m) are taken **From material**, which means that the values are taken from the boundary material.
- To specify other values for any of these properties, select **User defined** from the list and then enter a value or expression. T is the current temperature, which can be a value specified as a model input or the temperature from a heat transfer interface. The definition of the temperature field is in the **Model Inputs** section.

Porous Media

When **Porous media** is selected, right-click to add a Porous Media subnode.

Archie's Law

When **Archie's law** is selected, right-click to add an Archie's Law subnode.

ELECTRIC FIELD

The default **Relative permittivity** ε_r (unitless) for the media is used **From material** and defined on the shell domain. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the permittivity and then enter values or expressions in the field or matrix.

MAGNETIC FIELD

Specify the **Constitutive relation** that describe the macroscopic properties of the medium (relating the magnetic flux density $\mathbf{B}$ and the magnetic field $\mathbf{H}$) and the applicable material properties, such as the relative permeability.

Select a **Constitutive relation**—**Relative permeability** (the default), **HB curve**, **Magnetic losses**, **Remanent flux density**, or **Magnetization**.


Note

The equation for the selected constitutive relation displays under the list. For all options, the default uses values **From material**, or select **User defined** to enter a different value or expression.

- Select **Relative permeability** μ_r (unitless) to use the constitutive relation $\mathbf{B} = \mu_0\mu_r\mathbf{H}$. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** and enter values or expressions in the field or matrix.
- Select **HB curve** $|\mathbf{H}|$ (SI unit: A/m) to use a curve that relates magnetic flux density $\mathbf{B}$ and the magnetic field $\mathbf{H}$ as $|\mathbf{H}| = f(|\mathbf{B}|)$.


Important

Do not select this option if using the **Induction Heating** interface. This option is not relevant for time harmonic modeling.

- Select **Magnetic losses** μ' and μ'' (unitless) to describe the relative permeability as a complex-valued quantity: $\mu_r = \mu' + i\mu''$, where μ' and μ'' are the real and imaginary parts, respectively.


Note

This option is not available for the **Magnetic Fields, No Currents** interface.

- Select **Remanent flux density** $\mathbf{B}_r$ (SI unit: T) to use the constitutive relation $\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$, where $\mathbf{B}_r$ is the remanent flux density (the flux density when no magnetic field is present).
  - The default relative permeability μ_r (unitless) uses values **From material**. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the relative permeability and enter another value or expression in the field or matrix.
  - Enter **x** and **y** components for the for the remanent flux density $\mathbf{B}_r$. For 3D models, enter **x**, **y**, and **z** components.

- Select **Magnetization M** (SI unit: A/m) to use the constitutive relation $\mathbf{B} = \mu_0\mathbf{H} + \mu_0\mathbf{M}$. Enter **x** and **y** components. For 3D models, enter **x**, **y**, and **z** components.

External Current Density

The **External Current Density** feature adds an externally generated current density $\mathbf{J}_e$, which appears on the right-hand side of the equation that the **Magnetic Fields** interface defines.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define an external current density.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

EXTERNAL CURRENT DENSITY

Enter a value or expression for each component of the **External current density** $\mathbf{J}_e$ (SI unit: A/m^2).

Velocity (Lorentz Term)

The **Velocity (Lorentz term)** feature adds velocity **v**. The external current is equal to $\sigma\mathbf{v} \times \mathbf{B}$. This feature is only valid as follows

- When solving for both the electric potential and the magnetic vector potential using the **Magnetic and Electric Fields** interface.
- In 2D and 2D axisymmetry when solving for only the out-of-plane component of the magnetic vector potential.

Important	To use the velocity feature correctly requires deep physical insight. In situations when the moving domain is of bounded extent in the direction of the motion or material properties vary in this direction or it contains magnetic sources that also move, the Lorentz term must not be used.

An operational definition of when it can be used is that the moving domain should only contain an induced magnetic source (magnetization + eddy currents) that has to be stationary with respect to the motion. Thus, it cannot be used for modeling projectiles

of finite length or projectiles containing magnets. It can be used to model conductive, homogeneous spinning disks (magnetic brakes), magnets over a moving infinite homogenous plane (maglev trains), a flow of homogeneous conducting fluid past a magnet (liquid metal pumps, Hall generators/thrusters).

If in doubt, contact COMSOL Technical Support
www.comsol.com/support

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the velocity.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

VELOCITY (LORENTZ TERM)

Select **User defined** to enter the components for the **Velocity** vector **v** (SI unit: m/s) or, if present, select any velocity field—**Velocity field (spf/pf1)**, for example—defined in the model.

For example, using the velocity field is useful when coupling to the velocity field of a fluid for a magnetohydrodynamic model.

Initial Values

The **Initial Values** feature adds an initial value for the magnetic vector potential A that can serve as an initial value for a transient simulation or as an initial guess for a nonlinear solver.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define an initial value.

INITIAL VALUES

Enter values or expressions for the initial value of the **Magnetic vector potential A** (SI unit: Wb/m). The default is 0.

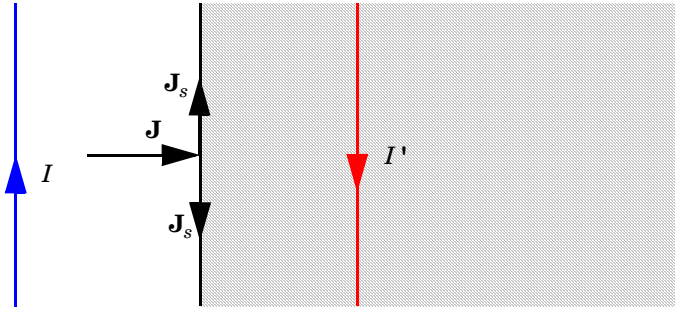
Magnetic Insulation

The **Magnetic Insulation** node is the default boundary condition for the **Magnetic Fields** interface and adds a boundary condition that sets the tangential components of the magnetic potential to zero at the boundary $\mathbf{n} \times \mathbf{A} = 0$.


Tip

Magnetic insulation is a special case of the magnetic potential boundary condition that sets the tangential component of the magnetic potential to zero.

It is used for the modeling of a lossless metallic surface, for example a ground plane or as a symmetry type boundary condition. It imposes symmetry for magnetic fields and "magnetic currents." In the transient and time harmonic formulations it also imposes antisymmetry for electric fields and electric currents. It supports induced electric surface currents and thus any prescribed or induced electric currents (volume, surface, or edge currents) flowing into a perfect electric conductor boundary is automatically balanced by induced surface currents.



The magnetic insulation boundary condition is used on exterior and interior boundaries representing the surface of a lossless metallic conductor or (on exterior boundaries) representing a symmetry cut. The shaded (metallic) region is not part of the model but still carries effective mirror images of the sources. Note also that any current flowing into the boundary is perfectly balanced by induced surface currents. The tangential vector potential (and electric field) vanishes at the boundary.

BOUNDARY SELECTION


Note

For the default node no user selection is required. **All boundaries** is automatically selected, it applies to all the *external* boundaries.

See Also

Magnetic Fields Interface Advanced Features for more domain features and boundary conditions.

When additional nodes are added, from the **Selection** list choose the boundaries to define the magnetic insulation.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Interface Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

Magnetic Field

The **Magnetic Field** feature adds a boundary condition for specifying the tangential component of the magnetic field at the boundary:

$$\mathbf{n} \times \mathbf{H} = \mathbf{n} \times \mathbf{H}_0$$

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to specify the magnetic field.

PAIR SELECTION

If **Magnetic Field** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

MAGNETIC FIELD

Enter the value or expression for the **Magnetic Field** $\mathbf{H}_0$ (SI unit: A/m) coordinates.

See Also

Magnetic Fields Interface Advanced Features for more boundary conditions.

Surface Current

The **Surface Current** feature adds a boundary condition for a surface current density $\mathbf{J}_s$:

$$-\mathbf{n} \times \mathbf{H} = \mathbf{J}_s$$
$$\mathbf{n} \times (\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{J}_s$$

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to specify a surface current.

PAIR SELECTION

If **Surface Current** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

SURFACE CURRENT

Enter values or expressions for the **Surface current density** $\mathbf{J}_s$ (SI unit A/m) coordinates.


See Also

Magnetic Fields Interface Advanced Features for more boundary conditions.

Magnetic Potential

The **Magnetic Potential** feature adds a boundary condition for the magnetic vector potential:

$$\mathbf{n} \times \mathbf{A} = \mathbf{n} \times \mathbf{A}_0$$

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to specify the magnetic potential.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

PAIR SELECTION

If **Magnetic Potential** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

MAGNETIC POTENTIAL

Enter a value or expression for the **Magnetic vector potential** $\mathbf{A}_0$ (SI unit: Wb/m) coordinates.

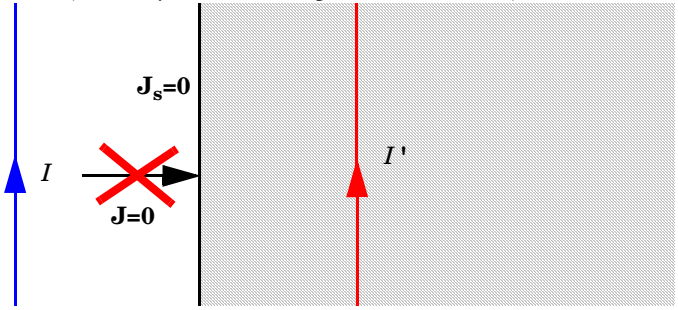
Magnetic Fields Interface Advanced Features for more boundary conditions.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

Perfect Magnetic Conductor

The **Perfect Magnetic Conductor** boundary condition $\mathbf{n} \times \mathbf{H} = 0$ is a special case of the surface current boundary condition that sets the tangential component of the magnetic field and thus also the surface current density to zero. On external boundaries, this can be interpreted as a "high surface impedance" boundary condition or used as a symmetry type boundary condition. It imposes symmetry for electric fields and electric currents. Electric currents (volume, surface, or edge currents) are not allowed to flow into a perfect magnetic conductor boundary as that would violate current conservation. On interior boundaries, the perfect magnetic conductor boundary condition literally sets the tangential magnetic field to zero which in addition to setting the surface current density to zero also makes the tangential magnetic vector potential (and in dynamics the tangential electric field) discontinuous.



The perfect magnetic conductor boundary condition is used on exterior boundaries representing the surface of a high impedance region or a symmetry cut. The shaded (high impedance) region is not part of the model but nevertheless carries effective mirror images

of the sources. Note also that any electric current flowing into the boundary is forbidden as it cannot be balanced by induced electric surface currents. The tangential magnetic field vanishes at the boundary. On interior boundaries, the perfect magnetic conductor boundary condition literally sets the tangential magnetic field to zero which in addition to setting the surface current density to zero also makes the tangential magnetic vector potential (and in dynamics the tangential electric field) discontinuous.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to model as perfect magnetic conductors.

PAIR SELECTION

If **Perfect Magnetic Conductor** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.


See Also

Magnetic Fields Interface Advanced Features for more boundary conditions.

Line Current (Out-of-Plane)


2D


2D Axi

This feature is available for 2D and 2D axially symmetric models.

Use the **Line Current (Out-of-Plane)** feature, selected from the **Points** menu, to specify a line current out of the modeling plane. In axially symmetric geometries this is the rotational direction, in 2D geometries this is the z-direction.

POINT SELECTION

From the **Selection** list, choose the points to add a line current.

LINE CURRENT (OUT-OF-PLANE)

Enter a value or expression for the **Out-of-plane current** I_0 (SI unit: A).

See Also

Magnetic Fields Interface Advanced Features for more point conditions.

Magnetic Fields Interface Advanced Features

For the AC/DC Module, several advanced features and subfeatures are available with this interface. In addition to the nodes described in The Magnetic Fields Interface, this section details these nodes and subnodes (listed in alphabetical order):

- Automatic Current Calculation
- Boundary Feed
- Coil Group Domain
- Edge Current
- Electric Insulation
- External Magnetic Vector Potential
- Gap Feed
- Gauge Fixing for A-field
- Impedance Boundary Condition
- Input
- Lumped Port
- Magnetic Point Dipole
- Magnetic Point Dipole (on Axis)
- Multi-Turn Coil Domain
- Output
- Reference Edge
- Reversed Current Direction
- Single-Turn Coil Domain

- Thin Low Permeability Gap
- Transition Boundary Condition


See Also

- Electric Point Dipole and Electric Point Dipole (on Axis) as defined for the **Electric Currents** interface
- Force Calculation as defined for the **Electrostatics** interface
- About Infinite Element Domains and Perfectly Matched Layers in the *COMSOL Multiphysics User's Guide*


Tip

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

Gauge Fixing for A-field


2D


2D Axi

For 2D and 2D axisymmetric models. Gauge fixing is available when vector (curl) shape functions are used, that is, when having in-plane dependent variables. The node is made available when **In-plane vector potential** or **Three-component vector potential** is selected from the **Components** section on The Magnetic Fields Interface settings window.


3D

For 3D models, and when applicable, right-click the **Ampère's Law** node to add a **Gauge-Fixing for A-field** feature. No additional settings are required to add the feature.

The **Gauge Fixing for A-Field** feature provides gauge fixing by adding an additional potential variable, ψ, and its associated conservation equation to the system. This is

often necessary to get a unique and numerically stable solution to the equation solving for the magnetic vector potential **A**.


See Also

Explicit Gauge Fixing/Divergence Constraint

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the gauge-fixing potential ψ.

GAUGE FIXING FOR A-FIELD

Enter a **Divergence condition variable scaling** ψ_0 (SI unit: A/m). The default value is 1, which means no scaling.

Multi-Turn Coil Domain

The **Multi-Turn Coil Domain** feature is a lumped model for a bundle of tiny wires tightly wound together, separated by an electrical insulator. In this scenario, the current in the domain flows only in the direction of the wires and is negligible in other directions. As a consequence, the settings for the **Electric Field** and **Magnetic Field** sections are the same as in Ampère's Law node, but the **Conduction Current** section is not available.

The material properties specified in these sections should be the homogenized electric and magnetic properties of the materials (insulator and wires) present in the coil domain. Right-click the node to add a Reference Edge subnode as required.


2D


2D Axi

In 2D and 2D axisymmetric models, the wires are assumed to be in the out-of-plane direction.


See Also

See Coil Domains in the modeling section to learn more about using this node.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the multi-turn coil domain.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COIL TYPE

 *3D*	This section is available for 3D models and is used to specify the coil geometry (the direction of the wires). Select a **Coil Type**—**Linear**, **Circular**, **Numeric**, or **User defined** and then go to the sections that follow. Also see Using Coils in 3D Models for more information about the options.

 *Important*	In three out of four models, an additional subfeature is needed to completely define the geometry.

Linear Coil Types

In a **Linear** coil, the wires are all parallel and straight lines and a **Reference Edge** subfeature is required. Right-click the **Multi-Turn Coil Domain** node to add a Reference Edge subfeature and select an edge or a group of co-linear edges. The direction of the wires and the coil length is taken to be the direction and the length of the edge(s).

To avoid unphysical currents, a Linear coil should be terminated on external boundaries.

Circular Coil Types

In a **Circular** coil, the wires are wound in circles around the same axis and a **Reference Edge** subfeature is required. Right-click the **Multi-Turn Coil Domain** node to add a

Reference Edge subfeature and select a group of edges forming a circle around the coil's axis. From the selected edge, the coil axis is computed and the direction of the wires is taken to be the azimuthal direction around the axis. The coil length used is simply the length of the edges; the best approximation is obtained when the radius of the edges is close to the average radius of the coil.

Numeric Coil Types

In a **Numeric** coil, the current flow is computed automatically in a **Coil Current Calculation** study step. An **Automatic Current Calculation** subfeature is needed to set up the problem.


See Also

Computing Coil Currents

User Defined Coil Types

Select **User defined** to manually specify the direction of the wires as a vector field and the length of the coil. Then enter values or expressions in the matrix for the **Coil current flow** (unitless) for **x**, **y**, and **z**. Enter a **Coil length** l_{coil} (SI unit: m).

MULTI-TURN COIL DOMAIN


See Also

- Coil Excitation
- Using Coils in 3D Models

Coil Name

Enter a **Coil name**. This name is appended to the global variables (current, voltage) defined by this coil, and it is used to identify the coil in a **Coil Current Calculation** study step.

Coil Conductivity

Enter a **Coil conductivity** σ_{coil} (SI unit: S/m). The default value is approximately the conductivity for copper, $6 \cdot 10^7$ S/m. This parameter represents the conductivity of the metal wires forming the coil. This is not the domain's bulk conductivity, that is instead set to zero, according to the lumped model of a bundle of wires.

Number of Turns

Enter the **Number of turns** N. the default is 10. This is the number of tiny wires constituting the coil. With the same current applied, more turns create a higher current density, but the total coil resistance increases as well.

Coil Wire Cross-Section Area

Enter a **Coil wire cross-section area** a_{coil} (SI unit: m^2). The default value is 10^{-6} m^2. This is the cross-section area of the individual wire in the bundle. It is used to compute the lumped resistance of the coil.

Coil Excitation

Select a **Coil excitation**—**Current**, **Voltage**, **Circuit (voltage)**, or **Circuit (current)**.

- If **Current** is selected, enter a **Coil current** I_{coil} (SI unit: A). The default is 1 A.
- If **Voltage** is selected, enter a **Coil voltage** V_{coil} (SI unit: V). The default is 1 V.
- If **Circuit (current)** is selected, it works similarly to the **Current** excitation, but in this case the inputs are provided by a circuit connection.
- If **Circuit (voltage)** is selected, it works similarly to the **Voltage** excitation, but in this case the inputs are provided by a circuit connection.


2D


2D Axi

For 2D and 2D axisymmetric models, **Power** is also available as a **Coil excitation** option. If **Power** is selected, enter a **Coil power** P_{coil} (SI unit: W). The default value is 1 W.


Note

Selecting **Power** makes the problem nonlinear. For more information see Power Excitation.

Single-Turn Coil Domain

The **Single-Turn Coil Domain** node models a conductive (for example, metallic) domain subject to a lumped excitation, such as voltage or current. The excitation specified is translated into a conduction current flowing in the domain. The **Single-Turn Coil Domain** feature is a contributing feature, that is, it is applied on top of an Ampère's Law feature that provides the material model for the domain.

In 2D and 2D axisymmetric models, the direction of the applied electric field is assumed to be out-of-plane. The settings specify how to compute the electric field and the relative current density.

The approach for 3D models is different than for 2D and 2D axisymmetric models. An additional dependent variable V_{coil}, with the dimension of an electric potential, is defined in the domain and the current continuity equation is added to the system of equations. The problem solved in the domain becomes effectively an A-V formulation of Maxwell's equation and current conservation, similar to the one used in the **Magnetic and Electric Fields** interface.

For 3D models, right-click the **Single-Turn Coil Domain** node to add Gap Feed, Boundary Feed, and Ground subnodes to define the geometry of the coil.

Tip

A typical setup for a 3D model is to add a **Gap Feed**, if the coil is closed in a loop, or a **Boundary Feed** and a **Ground**, if the coil is open. In order to avoid unphysical current distribution in the **Boundary Feed** and **Ground** case, the boundary conditions should be applied on external boundaries.

See Also

See Coil Domains in the modeling section to learn more about using this node.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the single-turn coil domain.

SINGLE-TURN COIL DOMAIN (2D AND 2D AXISYMMETRIC MODELS)

2D

2D Axi

In 2D and 2D axisymmetric models, the direction of the applied electric field is assumed to be out-of-plane. These settings specify how to compute the electric field and the relative current density.

Coil Name

Enter a **Coil name**. This name is appended to the global variables (current, voltage) defined by this coil, and it is used to identify the coil in a **Coil Current Calculation** study step.

Coil Excitation

Select a **Coil excitation**—**Current**, **Voltage**, **Circuit (voltage)**, **Circuit (current)**, or **Power**.

- If **Current** is selected, also enter a **Coil current** I_{coil} (SI unit: A). The default value is 1 A.
- If **Voltage** is selected, enter a **Coil potential** V_{coil} (SI unit: V). The default value is 1 V.
- If **Circuit (current)** is selected, it works similarly to the **Current** excitation, but in this case the inputs are provided by a circuit connection.
- If **Circuit (voltage)** is selected, it works similarly to the **Voltage** excitation, but in this case the inputs are provided by a circuit connection.
- If **Power** is selected, enter a **Coil power** P_{coil} (SI unit: W). The default value is 1 W.


Note

Selecting **Power** makes the problem nonlinear. For more information see Power Excitation.

SINGLE-TURN COIL DOMAIN (3D MODELS)


3D

For 3D models, the parameters **Coil conductivity** and **Coil relative permittivity** define the material model to be used with the current continuity equation. Normally, these parameters are the same used in the Ampère's Law feature active in the domain.

Coil Conductivity

Select a **Coil conductivity** σ_{coil} (SI unit: S/m)—**From material** or **User defined**. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the coil conductivity, and then enter values or expressions in the field or matrix.

Coil Relative Permittivity

Select a **Coil relative permittivity** $\varepsilon_{\text{r,coil}}$ (unitless)— **From material** or **User defined**. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the

characteristics of the coil relative permittivity, and then enter values or expressions in the field or matrix.

Gap Feed

Right-click the Single-Turn Coil Domain node to add the **Gap Feed** subnode. **Gap Feed** is used to apply a discontinuity in the coil potential across a boundary. This feature must be applied to an internal boundary in the coil domain, and is typically used to excite a closed loop.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define the gap feed. The default selects **All boundaries**.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty.

SINGLE-TURN COIL DOMAIN

See Single-Turn Coil Domain for all settings.

See Also

See Using Coils in 3D Models in the modeling section to learn more about this node.

Boundary Feed

Right-click the Single-Turn Coil Domain node to add the **Boundary Feed** subnode. The **Boundary Feed** feature acts on the coil potential and must be applied to an external boundary of the coil domain.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define the gap feed. The default selects **All boundaries**.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty.

SINGLE-TURN COIL DOMAIN

See Single-Turn Coil Domain for all settings.


See Also

See Using Coils in 3D Models in the modeling section to learn more about this node.

Ground

Right-click the Single-Turn Coil Domain node to add the **Ground** subnode. The **Ground** subfeature enforces the condition $V_{coil} = 0$ on a boundary.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define the ground. The default selects **All boundaries**.


See Also

See Using Coils in 3D Models in the modeling section to learn more about this node.

Reference Edge

After selecting **Linear** or **Circular** as the **Coil type** for the **Multi-Turn Coil Domain** feature, right-click the Multi-Turn Coil Domain node to add the **Reference Edge** subnode.

The **Reference Edge** subfeature is used with **Linear** coils where the wires are all parallel and straight lines. Select an edge or a group of co-linear edges. The direction of the wires and the coil length is taken to be the direction and the length of the edge(s). To avoid unphysical currents, a **Linear** coil should be terminated on external boundaries.

The **Reference Edge** subfeature is used with a **Circular** coil where the wires are wound in circles around the same axis. Select a group of edges forming a circle around the coil's axis. From the selected edge, the coil axis is computed and the direction of the wires is taken to be the azimuthal direction around the axis. The coil length used is simply the length of the edges; the best approximation is obtained when the radius of the edges is close to the average radius of the coil.

EDGE SELECTION

From the **Selection** list, choose the edges to define the reference edge. The default selects **All edges**.


See Also

See Using Coils in 3D Models in the modeling section to learn more about this node.

Automatic Current Calculation


3D

This subnode is available for 3D models and when the **Coil Type** selected is **Circular**.

Right-click the Multi-Turn Coil Domain to add an **Automatic Current Calculation** subnode. This subfeature is needed to set up the automatic computation of the current flow in the coil domain. The boundary conditions for the current calculation study are specified using the Electric Insulation, Input, and Output subnodes.

To complete the setup for the computation of the coil direction, a **Coil Current Calculation** study step must be added to the study.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the automatic current calculation.

AUTOMATIC CURRENT CALCULATION

The parameter **Off-diagonal scaling** is the scaling used to stabilize the eigenvalue problem. The default value is 0.1 and should be valid for most cases.


See Also

- Using Coils in 3D Models
- Computing Coil Currents
- Solver Features in the *COMSOL Multiphysics Reference Guide*

Electric Insulation

Right-click the Automatic Current Calculation node to add a **Electric Insulation** subnode. This is the default boundary condition. Apply it to the boundaries delimiting the coil domain, which then constrains the coil wires to be parallel to the boundary.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define the electric insulation for the automatic current calculation.


See Also

- Using Coils in 3D Models
- Computing Coil Currents
- Solver Features in the *COMSOL Multiphysics Reference Guide*

Input

Right-click the Automatic Current Calculation node to add an **Input** subnode and specify the boundaries where the wires enter the domain. The wire direction is forced to be orthogonal to the boundary. Used in combination with the Output node, it also defines the direction of the current flow (from **Input** to **Output**).

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define the input for the automatic current calculation.


See Also

- Using Coils in 3D Models
- Computing Coil Currents
- Solver Features in the *COMSOL Multiphysics Reference Guide*

Output

Right-click the Automatic Current Calculation node to add an **Output** subnode. and specify the boundaries where the wires exit the domain. The wire direction is forced to be orthogonal to the boundary. Used in combination with the Input node, it also defines the direction of the current flow (from **Input** to **Output**).

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define the output for the automatic current calculation.

See Also

- Using Coils in 3D Models
- Computing Coil Currents
- Solver Features in the *COMSOL Multiphysics Reference Guide*

Coil Group Domain

2D

2D Axi

The **Coil Group Domain** feature is available for 2D and 2D axisymmetric models.

The **Coil Group Domain** node adds an externally generated current density to the right-hand side of the equation that the **Magnetic Fields** interface defines. This current density is calculated in three different ways, depending on whether a fixed current in each coil turn, a total voltage drop across the coil, or a fixed power into the coil is specified. Right-click to add Reversed Current Direction and Harmonic Perturbation nodes.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the coil group domain.

COIL GROUP DOMAIN

Enter a **Coil name**. This name is appended to the global variables (current, voltage) defined by this coil, and it is used to identify the coil in a **Coil Current Calculation** study step.

Select a **Coil excitation**—**Current**, **Voltage**, **Power**, **Circuit (voltage)**, or **Circuit (current)**.

- If **Current** is selected, also enter a **Coil current** I_{coil} (SI unit: A). The default is 1 A.
- If **Voltage** is selected, also enter a **Coil potential** V_{coil} (SI unit: V). The default is 1 V.

- If **Circuit (current)** is selected, it works similarly to the **Current** excitation, but in this case the inputs are provided by a circuit connection.
- If **Circuit (voltage)** is selected, it works similarly to the **Voltage** excitation, but in this case the inputs are provided by a circuit connection.
- If **Power** is selected, enter a **Coil power** P_{coil} (SI unit: W). The default value is 1 W.


Note

Selecting **Power** makes the problem nonlinear. For more information see Power Excitation.


See Also

- Coil Domains
- Coil Excitation

Reversed Current Direction

Right-click the Coil Group Domain node to add the **Reversed Current Direction** node.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the reversed current direction.


See Also

Coil Domains

Harmonic Perturbation

Right-click the Coil Group Domain node to add the **Harmonic Perturbation** feature. Use a **Harmonic Perturbation** node to add harmonic perturbation to the right-hand-side contributions of the parent node. The perturbation of the contribution is entered in

the settings window, which is used when solving for a Frequency-Domain, Perturbation study type.


See Also

Harmonic Perturbation, Prestressed Analysis, and Small-Signal Analysis in the *COMSOL Multiphysics User's Guide*

DOMAIN SELECTION

From the **Selection** list, choose the domains to impose the harmonic perturbation.

HARMONIC PERTURBATION

Enter a **Coil current** I_{coil} (SI unit: A). The default is 0.

Lumped Port

Use the **Lumped Port** condition to apply a uniform electric field between two metallic boundaries. The excitation at the port can be expressed as a voltage or as a current, or via the connection to a circuit interface. The use of this feature is justified when the distance between the metallic boundaries is much smaller than the wavelength of the electromagnetic radiation, or for stationary studies.


Note

See S-Parameters and Ports and Lumped Ports with Voltage Input for more information.

The geometry of the port is specified by the **Type of Port**. A **Uniform** lumped port applies a constant electric field between the metallic electrodes. A **Coaxial** lumped port applies a radial electric field between two concentric circular metallic boundaries. For these two cases, the dimension of the port is computed automatically by analyzing the geometry. Generally, select **User defined** to manually specify the direction between the

lumped port terminals a_h and the dimensions of the port: h_port (Height of lumped port) and w_port (Width of lumped port).

 *2D*	For 2D models, a Uniform port applies an in-plane electric field. Therefore, the Uniform value is available for the Port Type parameter only if the in-plane vector potential is solved for, according to the settings in the Physics interface node. If the Components selected are Out-of-plane vector potential, the Uniform lumped port is not available.

 *2D Axi*	For 2D axisymmetric models, the radial direction is in-plane with respect to the geometry. Therefore, the Uniform value is available for the Port Type parameter only if the in-plane vector potential is solved for, according to the settings in the Physics interface node. If the Components selected are Out-of-plane vector potential, the Coaxial lumped port is not available.

 *2D*  *2D Axi*	For 2D and 2D axisymmetric models, if the Components selected in the Physics interface is Out-of-plane vector potential, the dimension of the port cannot be obtained by analyzing the geometry and must be specified manually by the user. Therefore, User defined is the only option available for the Port Type parameter.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to specify the lumped port.

PAIR SELECTION

If **Lumped Port** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

PORT PROPERTIES

Enter a unique **Port Name**. It is recommended to use a numeric name as it is used to define the elements of the S-parameter matrix and numeric port names are also required for port sweeps and Touchstone file export.

Type of Port

The geometry of the port is specified by the **Type of Port**. A **Uniform** lumped port applies a constant electric field between the metallic electrodes. A **Coaxial** lumped port applies a radial electric field between two concentric circular metallic boundaries. For these two cases, the dimension of the port is computed automatically by analyzing the geometry.

 *3D*	Select a **Type of Port**—**Uniform**, **Coaxial**, or **User defined**.

 *2D*	The **Type of Port** options available depend on the **Components** selected on the physics interface—**Out-of-plane vector potential** (the default), **In-plane vector potential**, or **Three-component vector potential.** • If **Out-of-plane vector potential** is selected, the **Type of Port** is **User defined**. • If **In-plane vector potential** or **Three-component vector potential** is selected, choose a **Type of Port**—**Uniform** or **User defined**.

 *2D Axi*	The **Type of Port** options available depend on the **Components** selected on the physics interface—**Out-of-plane vector potential** (the default), **In-plane vector potential**, or **Three-component vector potential.** • If **Out-of-plane vector potential** is selected, the **Type of Port** is **User defined**. • If **In-plane vector potential** or **Three-component vector potential** is selected, choose a **Type of Port**—**Coaxial** or **User defined**.

Select **User defined** for non uniform ports, for example, a curved port and enter values or expressions in the fields for these additional settings:

- **Height of lumped port** h_{port} (SI unit: m)
- **Width of lumped port** w_{port} (SI unit: m)
- The coordinates based on space dimension for the **Direction between lumped port terminals** $\mathbf{a}_{\text{h}}$

Terminal Type

For all **Types of ports**, select a **Terminal type**—a **Cable** port for a voltage driven transmission line, a **Current** driven port, or a **Circuit** port.

Wave Excitation at this Port


Note

This section is available if **Cable** is selected as the **Terminal type**.

Select **On** or **Off** from the **Wave excitation at this port** list to set whether it is an inport or a listener port. If **On** is selected, enter a **Voltage** V_0 (SI unit: V), and **Port phase** θ (SI unit: rad).


Tip

It is only possible to excite one port at a time if the purpose is to compute S-parameters. In other cases, for example, when studying microwave heating, more than one inport might be wanted, but the S-parameter variables cannot be correctly computed so if several ports are excited, the S-parameter output is turned off.

SETTINGS


Note

This section is available if **Cable** or **Current** is selected as the **Terminal type**.

- If **Cable** is selected, enter the **Characteristic impedance** Z_{ref} (SI unit: Ω).
- If **Current** is selected, enter a **Terminal current** I_0 (SI unit: A)

Edge Current

Use the **Edge Current** feature to specify a line current along one or more edges.

EDGE SELECTION

From the **Selection** list, choose the edges to apply an edge current.

EDGE CURRENT

Specify the **Edge current** I_0 (SI unit: A).

External Magnetic Vector Potential


Note

This feature is only available when solving a problem with a background magnetic vector potential (**Reduced field** is selected from the **Solve for** list under **Background Field** on the interface settings window).

The **External Magnetic Vector Potential** boundary condition forces the reduced magnetic vector potential to be zero on the boundary, or, equivalently, forces the total field to be equal to the background field. Apply this boundary condition on external boundaries that are at a distance far enough from the system so that its effect on the background field is negligible.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to specify the external magnetic vector potential.

Impedance Boundary Condition

The **Impedance Boundary Condition** provides a boundary condition that is useful at boundaries where the electromagnetic field penetrates only a short distance outside the boundary.

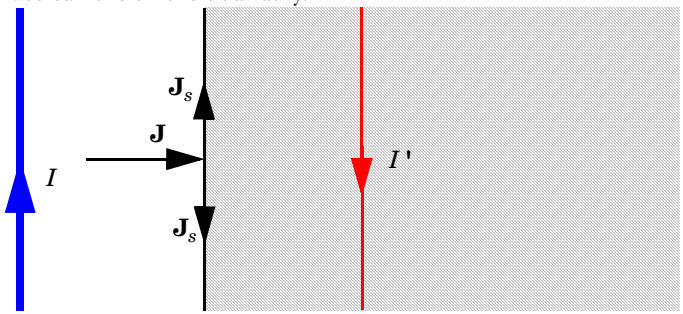
$$\sqrt{\frac{\mu_0\mu_r}{\varepsilon_0\varepsilon_r - j\sigma/\omega}}\mathbf{n}\times\mathbf{H} + \mathbf{E} - (\mathbf{n}\cdot\mathbf{E})\mathbf{n} = (\mathbf{n}\cdot\mathbf{E}_s)\mathbf{n} - \mathbf{E}_s$$

The boundary condition approximates this penetration to avoid the need to include another domain in the model. The material properties that appear in the equation are those for the domain outside the boundary.

The skin depth (that is, the distance where the electromagnetic field has decreased by a factor e^{-1}) is for a good conductor

$$\delta = \sqrt{\frac{2}{\omega\mu\sigma}}$$

The impedance boundary condition is a valid approximation if the skin depth is small compared to the size of the conductor. The source electric field $\mathbf{E}_s$ can be used to specify a source surface current on the boundary.



The impedance boundary condition is used on exterior boundaries representing the surface of a lossy domain. The shaded (lossy) region is not part of the model. The effective induced image currents are of reduced magnitude due to losses. Any current flowing into the boundary is perfectly balanced by induced surface currents as for the perfect electric conductor boundary condition. The tangential electric field is generally small but non zero at the boundary.


Model

Cold Crucible: Model Library path **ACDC_Module/Electromagnetic_Heating/cold_crucible**

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to specify the impedance boundary condition.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty. If a linear temperature relation is added for the conductivity, then define the source for the temperature T. From the **Temperature** list, select an existing temperature variable (from another physics interface) if available, or select **User defined** to define a value or expression for the temperature (SI unit: K) in the field that appears underneath the list.

MATERIAL TYPE

Select a **Material type**—**Solid**, **Non-solid**, or **From material**.

IMPEDANCE BOUNDARY CONDITION

The following material properties can be defined for the domain outside the boundary, which this boundary condition approximates. The default uses the values **From material.** Or select **User defined** to enter different values or expressions.

- **Relative permittivity**, ε_r (unitless)
- **Relative permeability**, μ_r (unitless)
- **Electrical conductivity**, σ (SI unit: S/m)

Based on space dimension, enter coordinate values or expressions for the **Source electric field** $\mathbf{E}_s$ (SI unit: V/m).

Transition Boundary Condition


Note

This feature is available with the **Frequency Domain** study type.

The **Transition Boundary Condition** is used on interior boundaries to model a sheet of a medium that should be geometrically thin but does not have to be electrically thin. It represents a discontinuity in the tangential electric field. Mathematically it is described by a relation between the electric field discontinuity and the induced surface current density:

$$\mathbf{J}_{s1} = \frac{(Z_S\mathbf{E}_{t1} - Z_T\mathbf{E}_{t2})}{Z_S^2 - Z_T^2}$$

$$\mathbf{J}_{s2} = \frac{(Z_S\mathbf{E}_{t2} - Z_T\mathbf{E}_{t1})}{Z_S^2 - Z_T^2}$$

$$Z_S = \frac{-j\omega\mu}{k}\frac{1}{\tan(kd)}$$

$$Z_T = \frac{-j\omega\mu}{k}\frac{1}{\sin(kd)}$$

$$k = \omega\sqrt{(\varepsilon + (\sigma/(j\omega)))\mu}$$

Where indices 1 and 2 refer to the different sides of the layer.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to specify the transition boundary condition.

MATERIAL TYPE

Select a **Material type**—**Solid**, **Non-solid**, or **From material**.

TRANSITION BOUNDARY CONDITION

The **Transition Boundary Condition** section has the following material properties for the thin layer, which this boundary condition approximates:

The defaults use the values **From material.** Or select **User defined** to enter different values or expressions.

- **Relative permittivity**, ε_r (unitless)
- **Relative permeability**, μ_r (unitless)
- **Electrical conductivity**, σ (SI unit: S/m)
- **Surface thickness**, d_s (Si unit: m)

Thin Low Permeability Gap

Use the **Thin Low Permeability Gap** boundary condition

$$\mathbf{n} \times (\mathbf{H}_1 - \mathbf{H}_2) = \nabla_t \times \frac{d}{\mu_0 \mu_r} \nabla_t \times \mathbf{A}$$

to model gaps filled with a material with zero conductivity such as air. This boundary condition is only applicable on interior boundaries and pair boundaries.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to model a thin low permeability gap.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty.

THIN LOW PERMEABILITY GAP

The default **Relative permeability** μ_r (unitless) is taken **From material**. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the

characteristics of the relative permeability, and then enter a value or expression in the field or matrix.

Enter a **Surface thickness** d_s (SI unit: m) for the gap.

Magnetic Point Dipole


2D

3D

Magnetic point dipoles are available with 2D and 3D models. See Magnetic Point Dipole (on Axis) for the 2D axisymmetric settings.

Apply a **Magnetic Point Dipole** $\mathbf{M}$ (SI unit: Am^2) to points. This represents the limiting case of when the cross-section area a of a circular current loop carrying uniform current I approaches zero at while maintaining the product between I and a. The dipole moment is a vector entity with the positive direction set by the curl of the current.

POINT SELECTION

From the **Selection** list, choose the points to add a magnetic point dipole.

DIPOLE SPECIFICATION

Select a **Dipole specification**—**Magnitude and direction** or **Dipole moment**.

DIPOLE PARAMETERS

If **Magnitude and direction** is selected under **Dipole Specification**, enter coordinates for the **Magnetic dipole moment direction** $\mathbf{n}_m$ and the **Magnetic dipole moment, magnitude** m (SI unit: Am^2).

If **Dipole moment** is selected under **Dipole Specification**, enter coordinates for the **Magnetic dipole moment** $\mathbf{m}$ (SI unit: Am^2).

Magnetic Point Dipole (on Axis)


2D Axi

This feature is available for 2D axisymmetric models. See Magnetic Point Dipole for settings for 2D and 3D models.

Use the **Magnetic Point Dipole (on Axis)** to apply a point dipole to points on a 2D axisymmetric model.

POINT SELECTION

From the **Selection** list, choose the points to add an magnetic point dipole.

MAGNETIC POINT DIPOLE (ON AXIS)

Enter a **Magnetic dipole moment in z direction** m_z (SI unit: Am^2).

The Magnetic Fields, No Currents Interface

The **Magnetic Fields, No Currents** interface ( ), found under the **AC/DC** branch ( ) of the **Model Wizard**, has the equations, boundary conditions, and point features for modeling magnetostatics, solving for the magnetic scalar potential. The main feature is the **Magnetic Flux Conservation** feature, which adds the equation for the magnetic potential and provides an interface for defining the material properties and the constitutive relation for the magnetic flux density. It is used when there are no currents.

When this interface is added, these default nodes are also added to the **Model Builder**—**Magnetic Fields, No Currents**, **Magnetic Flux Conservation, Magnetic Insulation** (the default boundary condition), and **Initial Values**. Right-click the **Magnetic Fields, No Currents** node to add other features that implement additional boundary conditions and point conditions.

Note	Except where described below, the settings windows are described for the **Magnetic Fields** and **Electrostatics** interfaces.

Model	• Magnetic Prospecting of Iron Ore Deposits: Model Library path **ACDC_Module/Magnetostatics/magnetic_prospecting** • Magnetic Signature of a Submarine: Model Library path **ACDC_Module/Magnetostatics/submarine**

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `mfnc`.

DOMAIN SELECTION

The default setting is to include **All domains** in the model to define the magnetic scalar potential and the equations that describe the potential field for magnetostatics without currents. To choose specific domains, select **Manual** from the **Selection** list.

BACKGROUND MAGNETIC FIELD

Select an option from the **Solve for** list—**Reduced field** or **Full field** (the default). If **Reduced field** is selected, specify a **Background magnetic field** $\mathbf{H}_b$ (SI unit: A/m). The total field used in the physics and equations are given by the sum of the reduced and background fields.

DEPENDENT VARIABLES

The dependent variable (field variable) is for the **Magnetic scalar potential** Vm. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. Select **Quadratic** (the default), **Linear, Cubic**, or **Quartic** for the **Magnetic scalar potential**. Specify the **Value type when using splitting of complex variables**—**Real** or **Complex** (the default).

See Also

- The Model Builder Show and Hide Physics Options
- Domain, Boundary, Point, and Pair Features for the Magnetic Fields, No Currents Interface
- Theory for the Magnetic Fields, No Currents Interface

Domain, Boundary, Point, and Pair Features for the Magnetic Fields, No Currents Interface

The Magnetic Fields, No Currents Interface has these domain, boundary, point, and pair conditions available.

Note

To obtain a unique solution, the value of the magnetic potential must be provided at least at one point. If the magnetic insulation boundary condition is used everywhere, the potential has to be fixed using a point condition.

About the Boundary Conditions

In magnetostatics the relevant interface condition between two domains 1 and 2 is

$$\mathbf{n}_2 \cdot (\mathbf{B}_1 - \mathbf{B}_2) = 0$$

This condition provides continuity of the normal component of the magnetic flux density and is automatically satisfied by the natural boundary condition for interior boundaries, which is

$$\mathbf{n} \cdot [(\mu_0 \nabla V_m - \mathbf{M})_1 - (\mu_0 \nabla V_m - \mathbf{M})_2] = -\mathbf{n} \cdot (\mathbf{B}_1 - \mathbf{B}_2) = 0$$

Available Features

These features are available for this interface and listed in alphabetical order. Also see Table 5-2 for a list of interior and exterior boundary conditions.

- External Magnetic Flux Density
- Initial Values
- Force Calculation (described for the **Electrostatics** interface)
- Magnetic Flux Conservation
- Magnetic Flux Density
- Magnetic Insulation (the default boundary condition)
- Magnetic Potential
- Magnetic Shielding
- Periodic Condition
- Sector Symmetry (described for the **Electric Currents** interface)
- Thin Low Permeability Gap
- Zero Magnetic Scalar Potential

See Also

In the *COMSOL Multiphysics User's Guide*:

- About Infinite Element Domains and Perfectly Matched Layers
- Continuity on Interior Boundaries
- Identity and Contact Pairs
- Specifying Boundary Conditions for Identity Pairs


Important

The links to the features described in the *COMSOL Multiphysics User's Guide* do not work in the PDF, only from within the online help.


Tip

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

Table 5-1 lists the interior and exterior boundaries available with this interface.

TABLE 5-2: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS FOR THE MAGNETIC FIELDS, NO CURRENTS INTERFACE

FEATURE	INTERIOR	EXTERIOR
Magnetic Flux Density		x
Magnetic Insulation	x	x
Magnetic Potential		x
Magnetic Shielding	x	x
Periodic Condition		x
Thin Low Permeability Gap	x	
Zero Magnetic Scalar Potential		x

Magnetic Flux Conservation

The **Magnetic Flux Conservation** feature adds Equation 5-5 above (or a similar equation depending on the selected constitutive relation) for the magnetic potential and provides an interface for defining the constitutive relation and the relevant material properties (for example, the relative permeability).

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the magnetic potential and the equation that describes the magnetic potential field.

MODEL INPUTS

This section has field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty. If a linear temperature relation is added for the conductivity, then the source can be defined for the temperature T. From the **Temperature** list, select an existing temperature variable (from

another physics interface) if available, or select **User defined** to define a value or expression for the temperature (SI unit: K).

MATERIAL TYPE

Select a **Material type**—**Solid**, **Non-solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems the model includes.

MAGNETIC FIELD


Note

See the settings for Magnetic Field under Ampère's Law for the **Magnetic Fields** interface, except for these differences:

- **Magnetic losses** is not an option for this interface.
- Select **BH curve** (instead of **HB curve**) to use a curve that relates magnetic flux density $\mathbf{B}$ and the magnetic field $\mathbf{H}$ as $|\mathbf{B}| = f(|\mathbf{H}|)$. To define $|\mathbf{B}|$, select **From material** (the default) to use the value from the material or select **User defined** to specify a value or expression for the magnitude of the magnetic flux density in the field that appears.

Initial Values

The **Initial Values** feature adds an initial value for the magnetic scalar potential that can serve as an initial guess for a nonlinear solver.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define an initial value.

INITIAL VALUES

Enter a value or expression for the initial value of the **Magnetic scalar potential** V_{m}. The default value is 0 A.

Infinite Elements

The **Infinite Elements** node imposes a coordinate transformation to the selected domain that effectively moves one or more sides of the domain to infinity. Infinite elements are

used for the modeling of open boundary problems. A default Magnetic Flux Conservation node is also added.

Magnetic Insulation

The **Magnetic Insulation** feature for the **Magnetic Fields, No Currents** interface provides magnetic insulation using the following boundary condition, which sets the normal component of the magnetic flux density to zero $\mathbf{n} \cdot \mathbf{B} = 0$.Magnetic insulation is the default boundary condition.

This condition is useful at boundaries confining a surrounding region of air or to model symmetry cuts.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define magnetic insulation.

Magnetic Flux Density

The **Magnetic Flux Density** feature adds a boundary condition for the magnetic flux density. The following equation defines the normal component of the magnetic flux density using a magnetic flux vector B0:

$$\mathbf{n} \cdot \mathbf{B} = \mathbf{n} \cdot \mathbf{B}_0 \tag{5-1}$$

Using this boundary condition specify the normal component of the magnetic flux density at the boundary.

Alternatively, specify an inward (or outward) flux density using the following equation:

$$-\mathbf{n} \cdot \mathbf{B} = B_\mathrm{n} \tag{5-2}$$

Using this formulation, it is possible to specify the normal component of the magnetic flux density as a scalar.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define a magnetic flux density.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems the model includes.

MAGNETIC FLUX DENSITY

Select a **Type** of the boundary condition to use—**Inward flux density** (the default) or **Magnetic Flux density**:

- If **Inward flux density** is selected, it defines the boundary condition according to Equation 5-2. Enter a scalar value or expression for the normal component of the **Inward flux density** B_n (SI unit: T). A positive value represents an inward flux.
- If **Magnetic flux density** is selected, it defines the boundary condition according to Equation 5-1. Enter a value or expression for each component of the **Magnetic flux density** $\mathbf{B}_0$ (SI unit: T) in the corresponding fields.

Zero Magnetic Scalar Potential

The **Zero Magnetic Scalar Potential** feature provides a boundary condition that specifies a zero magnetic potential on the boundary $V_\mathrm{m} = 0$.

BOUNDARY OR POINT SELECTION

From the **Selection** list, choose the geometric entity (boundaries or points) to define a zero magnetic potential.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

External Magnetic Flux Density


Note

This feature is only available when solving a problem with a background magnetic flux density (**Reduced field** is selected from the **Solve for** list under **Background Field** on the interface settings window).

The **External Magnetic Flux Density** boundary condition forces the reduced magnetic flux density to be zero on the boundary, or, equivalently, forces the total field to be equal to the background field. Apply this boundary condition on external boundaries

that are at a distance far enough from the system so that its effect on the background field is negligible.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define an external magnetic flux density.

Magnetic Shielding

The **Magnetic Shielding** feature adds a boundary condition for magnetic shielding. It describes a thin layer of a permeable medium that shields the magnetic field. The boundary condition uses the following equation:

$$\mathbf{n} \cdot (\mathbf{B}_1 - \mathbf{B}_2) = -\nabla_t \cdot (\mu_0 \mu_r d_s \nabla_t V_m)$$

In this equation, ∇_t represents a tangential derivative (gradient), and d_s is the surface thickness.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define magnetic shielding.

PAIR SELECTION

If **Magnetic Shielding** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

MATERIAL TYPE

Select a **Material type**—**Solid**, **Non-solid**, or **From material**.

MAGNETIC SHIELDING

The default **Relative permeability** μ_r (unitless) uses values **From material**. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** and enter other values or expressions. For anisotropic material, the relative permeability is a tensor.

Enter a value or expression for the **Surface thickness** d_s (SI unit: m).

Thin Low Permeability Gap

The **Thin Low Permeability Gap** feature adds a boundary condition on an internal boundary which allows for a discontinuity in the magnetic scalar potential. Enter a

relative magnetic permeability μ_r for the thin layer material as well as a thickness d_s. The magnetic flux through this boundary is given by

$$\mathbf{n} \cdot \mathbf{B} = \mu_0 \mu_r \left(\frac{V_m^d - V_m^u}{d_s} \right)$$

where $V_m{}^u$ is the magnetic scalar potential on the upside of the boundary selection and $V_m{}^u$ is the magnetic scalar potential on the downside.

Use the thin low permeability gap boundary condition

$$(\mathbf{n} \cdot \mathbf{B})_1 = \frac{\mu_0 \mu_r}{d}(V_{m1} - V_{m2})$$

$$(\mathbf{n} \cdot \mathbf{B})_2 = \frac{\mu_0 \mu_r}{d}(V_{m2} - V_{m1})$$

to model a thin gap of a low permeable material such as air. The layer has the thickness d and the relative permeability μ_r.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define a thin low permeability gap.

PAIR SELECTION

If **Pair Thin Low Permeability Gap** is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

MODEL INPUTS

This section has field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty.

MATERIAL TYPE

Select a **Material type**—**Solid**, **Non-solid**, or **From material**.

THIN LOW PERMEABILITY GAP

The default **Relative permeability** μ_r (unitless) uses values **From material**. Select **User defined** to enter a different value or expression. Enter a value or expression for the **Surface thickness** d_s (SI unit: m).

The Rotating Machinery, Magnetic Interface

The **Rotating Machinery, Magnetic** interface ( ), found under the **AC/DC** branch ( ) of the **Model Wizard**, combines an out-of-plane magnetic fields (magnetic vector potential) formulation with a selection of predefined frames for prescribed rotation or rotation velocity. It is used for 2D models only.

Important	The interface only works correctly if the geometry is created as an assembly pair from individual composite objects for the rotor and stator parts, respectively. An identity pair must also be defined for the rotor-stator interface boundaries under the **Model>Definitions** node in the **Model Builder**.

Note	Only features unique to the **Rotating Machinery, Magnetic** interface are described in this section. Most features are described for The Magnetic Fields Interface and Magnetic Fields Interface Advanced Features.

Model	Generator in 2D: Model Library path **ACDC_Module/ Motors_and_Actuators/generator_2d**

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `rmm`.

DOMAIN SELECTION

The default setting is to include **All domains** in the model. To choose specific domains, select **Manual** from the **Selection** list.

BACKGROUND FIELD

Select an option from the **Solve for** list—**Full field** (the default) or **Reduced field**. If **Reduced field** is selected, specify a **Background magnetic vector potential** $\mathbf{A}_b$ (SI unit: Wb/m). The total field used in the physics and equations are given by the sum of the reduced and background fields.

COMPONENTS

Select the **Components**—**Out-of-plane vector potential** (the default), **In-plane vector potential**, or **Three-component vector potential**.

THICKNESS

Enter a value or expression for the global **Out-of-plane thickness** d (SI unit: m). The default value of 1 m is typically not representative for a thin domain. Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D models.

Use the Change Thickness (Out-of-Plane) node to define specific domains instead of a global setting for the thickness.

SWEEP SETTINGS

Select the **Activate terminal sweep** check box and enter a **Sweep parameter name** in the field. The default is `PortName`.

DEPENDENT VARIABLES

The dependent variable (field variable) is for the **Magnetic vector potential** A. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. Select a **Magnetic vector potential**—**Linear**, **Quadratic** (the default), or **Cubic**. Specify the **Value type when using splitting of complex variables**—**Real** or **Complex** (the default).

See Also

- The Model Builder Show and Hide Physics Options
- Domain and Shared Features for the Rotating Machinery, Magnetic Interface

Domain and Shared Features for the Rotating Machinery, Magnetic Interface

Because The Rotating Machinery, Magnetic Interface is a multiphysics interface, many features are shared with, and described for, other interfaces. Below are links to the boundary, edge, pair, and point features as indicated.

These domain features are described in this section:

- Initial Values
- Electric Field Transformation
- Prescribed Rotation
- Prescribed Rotational Velocity

These boundary, edge, pair, and point conditions are described for other interfaces, including the **Magnetic Fields**, **Magnetic Fields, No Currents**, and **Electrostatics** interfaces (listed in alphabetical order):

- Ampère's Law
- Change Thickness (Out-of-Plane)
- Coil Group Domain
- Electric Point Dipole
- Electric Point Dipole (on Axis)
- External Current Density
- External Magnetic Vector Potential
- Force Calculation
- Impedance Boundary Condition
- Infinite Elements
- Line Current (Out-of-Plane)
- Lumped Port
- Magnetic Field
- Magnetic Insulation
- Magnetic Point Dipole
- Magnetic Potential
- Magnetic Shielding
- Multi-Turn Coil Domain

- Perfect Magnetic Conductor
- Periodic Condition
- Sector Symmetry
- Single-Turn Coil Domain
- Surface Current
- Thin Low Permeability Gap
- Transition Boundary Condition
- Velocity (Lorentz Term)


See Also

In the *COMSOL Multiphysics User's Guide*:

- Continuity on Interior Boundaries
- Identity and Contact Pairs
- Specifying Boundary Conditions for Identity Pairs


Important

The links to the features described in the *COMSOL Multiphysics User's Guide* do not work in the PDF, only from within the online help.


Tip

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

Initial Values

The **Initial Values** feature adds an initial value for the magnetic vector potential.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define an initial value.

INITIAL VALUES

Enter a value or expression for the initial value of the **Magnetic vector potential A** (SI unit: Wb/m).

Electric Field Transformation

The **Electric Field Transformation** feature imposes suitable transformations to the electric field definitions in all domains depending on rotational velocity.

DOMAIN SELECTION


Note

For the default node no user selection is required. **All domains** is automatically selected.

When additional nodes are added, from the **Selection** list choose the domains to define the electric field transformation.

Prescribed Rotation

The **Prescribed Rotation** feature imposes a coordinate transformation to the selected domain that effectively rotates it a prescribed angle. It is used to model a rotating part.

DOMAIN SELECTION

From the **Selection** list, choose the domains to use prescribed rotation.

PRESCRIBED ROTATION

Enter the **Rotation angle** α_{rot} (SI unit: radians) and the **x** and **y** coordinates for the **Rotation axis base point** $\mathbf{r}_{\text{ax}}$ (SI unit: m).

Prescribed Rotational Velocity

The **Prescribed Rotational Velocity** feature imposes a coordinate transformation to the selected domain that effectively rotates it a prescribed angle that grows linearly with time. It is used to model a rotating part.

DOMAIN SELECTION

From the **Selection** list, choose the domains to use prescribed rotational velocity.

PRESCRIBED ROTATIONAL VELOCITY

Enter the **Revolutions per second** rps (SI unit: Hz) and the **X** and **Y** coordinates for the **Rotation axis base point** $\mathbf{r}_{\text{ax}}$ (SI unit: m).

Theory of Magnetic and Electric Fields

Quasi-static analysis of magnetic and electric fields is valid under the assumption that $\partial \mathbf{D}/\partial t = 0$.

In this section:

Maxwell's Equations

This implies that it is possible to rewrite Maxwell's equations in the following manner:

$$\nabla \times \mathbf{H} = \mathbf{J} = \sigma(\mathbf{E} + \mathbf{v} \times \mathbf{B}) + \mathbf{J}_e$$

$$\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t}$$

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

$$\nabla \cdot \mathbf{D} = \rho$$

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

Here $\mathbf{J}_e$ is an externally generated current density and $\mathbf{v}$ is the velocity of the conductor. The crucial criterion for the quasi-static approximation to be valid is that the currents and the electromagnetic fields vary slowly. This means that the dimensions of the structure in the problem need to be small compared to the wavelength.

Magnetic and Electric Potentials

Using the definitions of the potentials,

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

$$\mathbf{E} = -\nabla V - \frac{\partial \mathbf{A}}{\partial t}$$

and the constitutive relation $\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$, Ampère's law can be rewritten as

$$\sigma\frac{\partial \mathbf{A}}{\partial t} + \nabla \times (\mu_0^{-1}\nabla \times \mathbf{A} - \mathbf{M}) - \sigma\mathbf{v} \times (\nabla \times \mathbf{A}) + \sigma\nabla V = \mathbf{J}_e \tag{5-3}$$

The equation of continuity, which is obtained by taking the divergence of the above equation, adds the following equation:

$$\nabla \cdot \left(-\sigma\frac{\partial \mathbf{A}}{\partial t} + \sigma\mathbf{v} \times (\nabla \times \mathbf{A}) - \sigma\nabla V + \mathbf{J}_e\right) = 0 \tag{5-4}$$

Equation 5-3 and Equation 5-4 form a system of equations for the two potentials $\mathbf{A}$ and V.

Gauge Transformations

The electric and magnetic potentials are not uniquely defined from the electric and magnetic fields through

$$\mathbf{E} = -\frac{\partial \mathbf{A}}{\partial t} - \nabla V$$

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

Introducing two new potentials

$$\tilde{\mathbf{A}} = \mathbf{A} + \nabla\Psi$$

$$\tilde{V} = V - \frac{\partial \Psi}{\partial t}$$

gives the same electric and magnetic fields:

$$\mathbf{E} = -\frac{\partial \mathbf{A}}{\partial t} - \nabla V = -\frac{\partial(\tilde{\mathbf{A}} - \nabla\Psi)}{\partial t} - \nabla\left(\tilde{V} + \frac{\partial \Psi}{\partial t}\right) = -\frac{\partial \tilde{\mathbf{A}}}{\partial t} - \nabla\tilde{V}$$

$$\mathbf{B} = \nabla \times \mathbf{A} = \nabla \times (\tilde{\mathbf{A}} - \nabla\Psi) = \nabla \times \tilde{\mathbf{A}}$$

The variable transformation of the potentials is called a *gauge transformation*. To obtain a unique solution, choose the gauge, that is, put constraints on Ψ that make the solution unique. Another way of expressing this additional condition is to put a

constraint on $\nabla \cdot \mathbf{A}$. A vector field is uniquely defined up to a constant if both $\nabla \cdot \mathbf{A}$ and $\nabla \times \mathbf{A}$ are given. This is called *Helmholtz's theorem*.

One particular gauge is the *Coulomb gauge* given by the constraint: $\nabla \cdot \mathbf{A} = 0$.

Selecting a Particular Gauge

Important observations are that in the dynamic case $\mathbf{A}$ and V are coupled via the selected gauge. For a dynamic formulation, it is also possible to select a Ψ such that the scalar electric potential vanishes and only the magnetic vector potential has to be considered. The dynamic formulations (frequency domain and time dependent study types) of the Magnetic Fields interface are operated in this gauge as it involves only $\mathbf{A}$. The Magnetic and Electric fields interface in the AC/DC Module involves both $\mathbf{A}$ and V and is inherently ungauged for all study types. In the static limit, $\mathbf{A}$ and V are not coupled via the gauge selection and thus any gauge can be chosen for $\mathbf{A}$ when performing magnetostatic modeling.

The Gauge and the Equation of Continuity for Dynamic Fields

After eliminating the electric potential by choosing the appropriate gauge and disregarding the velocity term. The equation of continuity obtained by taking the divergence of Ampère's law reads:

$$\nabla \cdot \left(-\sigma \frac{\partial \mathbf{A}}{\partial t} + \mathbf{J}_e\right) = 0$$

It is clear that unless the electrical conductivity is uniform, the particular gauge used to eliminate V cannot be the Coulomb gauge as that would violate the equation of continuity and would thereby also violate Ampère's law.

Explicit Gauge Fixing/Divergence Constraint

The AC/DC Module has a gauge fixing feature that is imposed by adding an extra scalar field variable ψ (not to be confused with Ψ used in the gauge transformation in the preceding section). The ψ field is used to impose a divergence constraint. In the most simple case, that is for magnetostatics, Ampère's law for the magnetic vector potential reads:

$$\nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) = \mathbf{J}_e$$

The equation for ψ is used to impose the *Coulomb gauge*: $\nabla \cdot \mathbf{A} = 0$. However, to get a closed set of equations, ψ must be able to affect the first equation and this is obtained by modifying the first equation to:

$$\nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) = \mathbf{J}_e + \nabla \psi$$

The additional term on the right-hand side can be seen as a Lagrange multiplier that not only imposes the Coulomb gauge but also eliminates any divergence in the externally generated current density, $\mathbf{J}_e$ and makes it comply with the current continuity inherent in Ampère's law.

The gauge fixing feature similarly imposes the Coulomb gauge also for the dynamic (frequency domain) study type in the Magnetic and Electric Fields interface.

For the dynamic (frequency domain and time dependent) study types for the Magnetic Fields interface the gauge is already determined so the gauge fixing feature is not allowed to impose the Coulomb gauge but reduces to help imposing current conservation. The first one is for the frequency domain study and the second one is for the time dependent study type.

$$\nabla \cdot \mathbf{J} = 0$$
$$\nabla \cdot (\sigma \mathbf{A}) = 0$$

The main benefit of using this kind of divergence constraint is improved numerical stability, especially when approaching the static limit when the inherent gauge deteriorates.

Ungauged Formulations and Current Conservation

Current conservation is inherent in Ampère's law and it is known that if current is conserved, explicit gauge fixing is not necessary as iterative solvers converge towards a valid solution. However, it is generally not sufficient for the source currents to be divergence free in an analytical sense as when interpolated on the finite element functional basis, this property is not conserved.

When using the Magnetic and Electric Fields interface the electric potential is used to state current conservation so unless nonphysical current sources are specified inside the computational domain current conservation is fulfilled.

When using the Magnetic Fields interface, current conservation is usually imposed either by the solver (for magnetostatics) or in the transient or time harmonic case by

the induced current density. The explicit gauge or divergence constraint can also help imposing current conservation as described in the preceding section.

Time-Harmonic Magnetic Fields

In the time-harmonic case, there is no computational cost for including the displacement current in Ampère's law (then called Maxwell-Ampère's law):

$$\nabla \times \mathbf{H} = \mathbf{J} = \sigma(\mathbf{E} + \mathbf{v} \times \mathbf{B}) + j\omega\mathbf{D} + \mathbf{J}^{\mathrm{e}}$$

In the transient case the inclusion of this term would lead to a second-order equation in time, but in the harmonic case there are no such complications. Using the definition of the electric and magnetic potentials, the system of equations becomes

$$-\nabla \cdot ((j\omega\sigma - \omega^2\varepsilon_0)\mathbf{A} - \sigma\mathbf{v} \times (\nabla \times \mathbf{A}) + (\sigma + j\omega\varepsilon_0)\nabla V - (\mathbf{J}^{\mathrm{e}} + j\omega\mathbf{P})) = 0$$

$$(j\omega\sigma - \omega^2\varepsilon_0)\mathbf{A} + \nabla \times (\mu_0^{-1}\nabla \times \mathbf{A} - \mathbf{M}) - \sigma\mathbf{v} \times (\nabla \times \mathbf{A}) + (\sigma + j\omega\varepsilon_0)\nabla V = \mathbf{J}^{\mathrm{e}} + j\omega\mathbf{P}$$

The constitutive relation $\mathbf{D} = \varepsilon_0\mathbf{E} + \mathbf{P}$ has been used for the electric field.

To obtain a particular gauge that reduces the system of equation, choose $\Psi = -jV/\omega$ in the gauge transformation. This gives

$$\tilde{\mathbf{A}} = \mathbf{A} - \frac{j}{\omega}\nabla V \qquad \tilde{V} = 0$$

When $\tilde{V}$ vanishes from the equations, only the second one is needed,

$$(j\omega\sigma - \omega^2\varepsilon_0)\tilde{\mathbf{A}} + \nabla \times (\mu_0^{-1}\nabla \times \tilde{\mathbf{A}} - \mathbf{M}) - \sigma\mathbf{v} \times (\nabla \times \tilde{\mathbf{A}}) = \mathbf{J}^{\mathrm{e}} + j\omega\mathbf{P}$$

Working with $\tilde{\mathbf{A}}$ is often the best option when it is possible to specify all source currents as external currents $\mathbf{J}^{\mathrm{e}}$ or as surface currents on boundaries.

Theory for the Magnetic Fields Interface

Simulation of magnetic fields is of interest when studying magnets, motors, transformers, and conductors carrying static or alternating currents.

The Magnetic Fields Interfaceis used for 3D, 2D in-plane, and 2D axisymmetric models. Unless you have a license for the AC/DC Module, only 2D modeling involving out-of-plane currents and axisymmetric modeling involving azimuthal currents are supported.

In this section:

- Magnetostatics Equation
- Frequency Domain Equation
- Transient Equation

See Also

For a deeper theoretical background to the magnetic vector potential used below, see the section starting with Maxwell's Equations.

Magnetostatics Equation

To derive the magnetostatic equation, start with Ampère's law for static cases $\nabla \times \mathbf{H} = \mathbf{J}$. The current is

$$\mathbf{J} = \sigma \mathbf{v} \times \mathbf{B} + \mathbf{J}^{\mathrm{e}}$$

where $\mathbf{J}^{\mathrm{e}}$ is an externally generated current density, and $\mathbf{v}$ is the velocity of the conductor.

Using the definitions of magnetic potential, $\mathbf{B} = \nabla \times \mathbf{A}$ and the constitutive relationship, $\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$, rewrite Ampère's law as

$$\nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) = \mathbf{J}^{\mathrm{e}}$$

which is the equation used in magnetostatics.


2D


2D Axi

The term involving the velocity only applies in the 2D and 2D axisymmetric formulations.

Frequency Domain Equation

To derive the time harmonic equation this physics interface solves, start with Ampère's law including displacement currents (then called Maxwell-Ampère's law) as these do not involve any extra computational cost in the frequency domain,

$$\nabla \times \mathbf{H} = \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} = \sigma \mathbf{E} + \sigma \mathbf{v} \times \mathbf{B} + \mathbf{J}^e + \frac{\partial \mathbf{D}}{\partial t}$$

Now assume time-harmonic fields and use the definitions of the fields,

$$\mathbf{B} = \nabla \times \mathbf{A}$$
$$\mathbf{E} = -j\omega \mathbf{A}$$

and combine them with the constitutive relationships $\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$ and $\mathbf{D} = \varepsilon_0 \mathbf{E}$ to rewrite Ampère's law as

$$(j\omega\sigma - \omega^2 \varepsilon_0)\mathbf{A} + \nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) = \mathbf{J}^e$$


2D


2D Axi

The term involving the velocity only applies in the 2D and 2D axisymmetric formulations.

Transient Equation

The transient equation this physics interface solves is Ampère's law, here illustrated with the constitutive relation $\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$.

$$\sigma\frac{\partial \mathbf{A}}{\partial t} + \nabla \times (\mu_0^{-1}\nabla \times \mathbf{A} - \mathbf{M}) - \sigma\mathbf{v} \times (\nabla \times \mathbf{A}) = \mathbf{J}_e$$


2D


2D Axi

The term involving the velocity only applies in the 2D and 2D axisymmetric formulations.

Theory for the Magnetic Fields, No Currents Interface

In magnetostatic problems where no electric currents are present, it is possible to formulate and solve the problem using a scalar magnetic potential. In a current-free region you have $\nabla \times \mathbf{H} = \mathbf{0}$. This implies that the magnetic scalar potential V_m can be defined from the relation $\mathbf{H} = -\nabla V_m$, which is analogous to the definition of the electric potential for static electric fields.

Using the constitutive relation $\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$, the equation $\nabla \cdot \mathbf{B} = 0$ becomes

$$-\nabla \cdot (\mu_0 \nabla V_m - \mu_0 \mathbf{M}) = 0 \tag{5-5}$$

The Magnetic Fields, No Currents Interface uses this equation for modeling of magnetostatics in the absence of electric currents.

In planar 2D the dynamic formulations also involves the thickness d in the z direction:

$$-\nabla \cdot d(\mu_0 \nabla V_m - \mu_0 \mathbf{M}) = 0$$

6

The Magnetic and Electric Fields Interface

In this chapter the Magnetic and Electric fields interface, found under the **AC/DC** branch () in the **Model Wizard**, is described.

The Magnetic and Electric Fields Interface

The **Magnetic and Electric Fields** interface ( ), found under the **AC/DC** branch ( ) of the **Model Wizard**, has the equations, boundary conditions, and external currents for modeling electric and magnetic fields, solving for the electric potential and magnetic vector potential.

Tip	If the conduction current in the modeled system is confined only in some regions (for example, a metallic coil) and most of the domains are non-conductive, it may be better to use the **Magnetic Fields** interface and apply the Coil Group Domain feature on the conductive regions, instead of using this interface.

Use the **Magnetic and Electric Fields** interface when the current continuity equation is needed everywhere in the simulation domain.

The main feature is the **Ampère's Law and Current Conservation** feature, which adds the equation for the electric potential and magnetic vector potential and provides an interface for defining the constitutive relations and their associated properties such as the relative permeability, relative permittivity, and electrical conductivity.

When this interface is added, these default nodes are also added to the **Model Builder**—**Ampère's Law and Current Conservation**, **Magnetic Insulation** (the default boundary condition for the magnetic vector potential), and **Initial Values**.

Right-click the **Magnetic and Electric Fields** node to add other features that implement, for example, boundary conditions and external currents.

Note	Except where described below, the majority of the settings windows are the same as for the **Magnetic Fields**, **Electrostatics**, and **Electric Currents** interfaces.


Model

- Magnetic Brake: Model Library path **ACDC_Module/Motors_and_Actuators/magnetic_brake**
- Inductance of a Power Inductor: Model Library path **ACDC_Module/Inductive_Devices_and_Coils/power_inductor**


Note

Use the Coil Domain features where possible. See Multi-Turn Coil Domain, Single-Turn Coil Domain, and Coil Group Domain for node information and Coil Domains for modeling information.

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `mef`.

DOMAIN SELECTION

The default setting is to include **All domains** in the model to define the electric potential, the magnetic vector potential, and the equations that describe the magnetic and electric fields. To choose specific domains, select **Manual** from the **Selection** list.

BACKGROUND FIELD

The only option available from the **Solve for** list is **Full field**.

COMPONENTS


2D


2D Axi

Select **Components**—**Out-of-plane vector potential** (the default), **In-plane vector potential**, or **Three-component vector potential**. From the practical viewpoint this choice is equivalent to deciding in what directions the electric current is allowed to flow (out-of-plane currents, in-plane currents or currents flowing in all three coordinate directions).

THICKNESS

2D

Enter a value or expression for the **Out-of-plane thickness** d. The default value of 1 unit length is typically not representative for a thin domain. Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D models.

SWEEP SETTINGS

Enter a **Reference impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω.

Select the **Activate terminal sweep** check box to switch on the sweep and invoke a parametric sweep over the terminals.

Select an option from the **Sweep on** list—**Terminals** or **Ports**.

Enter a **Sweep parameter name** to assign a specific name to the variable that controls the terminal number solved for during the sweep. The default is `PortName`. The generated lumped parameters are in the form of capacitance matrix elements. The terminal settings must consistently be of either fixed voltage or fixed charge type.

The lumped parameters are subject to **Touchstone file export**. Enter a file path or **Browse** for a file. Select an **Output format** for the Touchstone export—**Magnitude angle**, **Magnitude (dB) angle**, or **Real imaginary**.

When **Terminals** is selected under **Sweep on**, also select a **Parameter to export**—**Z** (the default), **Y**, or **S**.

See Also

Lumped Parameters

DEPENDENT VARIABLES

The dependent variables (field variables) are for the **Electric potential** V and **Magnetic vector potential** A. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button () and select **Discretization**. Select **Quadratic** (the default), **Linear, Cubic**, or **Quartic** for the **Electric potential** and **Magnetic**

vector potential. Specify the **Value type when using splitting of complex variables**—**Real** or **Complex** (the default).


See Also

- The Model Builder Show and Hide Physics Options
- Domain, Boundary, Edge, Point, and Pair Features for the Magnetic and Electric Fields Interface
- Theory for the Magnetic and Electric Fields Interface

Domain, Boundary, Edge, Point, and Pair Features for the Magnetic and Electric Fields Interface

The Magnetic and Electric Fields Interface has these domain, boundary, edge, point, and pair features.

About the Boundary Conditions

The **Magnetic and Electric Fields** interface boundary conditions are applied in a two-step procedure. This is because only some combinations of electric and magnetic boundary conditions are physically relevant whereas others may lead to nonphysical models and thus violate current conservation.


Tip

The basic steps for this are to right-click the **Magnetic and Electric Fields** node to add the *magnetic* boundary conditions. Then right-click these magnetic boundary condition nodes to add the *electric* boundary conditions as subnodes.

STEP 1: MAGNETIC BOUNDARY CONDITIONS

With no surface currents present the interface conditions

$$\mathbf{n}_2 \times (\mathbf{A}_1 - \mathbf{A}_2) = \mathbf{0}$$
$$\mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{0}$$

need to be fulfilled. Because the physics interface solves for $\mathbf{A}$, the tangential component of the magnetic potential is always continuous, and thus the first condition is automatically fulfilled. The second condition is equivalent to the natural boundary condition and is hence also fulfilled unless surface currents are explicitly introduced.

STEP 2: ELECTRIC BOUNDARY CONDITIONS

The relevant interface condition at interfaces between different media and interior boundaries is continuity; that is,

$$\mathbf{n}_2 \cdot (\mathbf{J}_1 - \mathbf{J}_2) = 0$$

which is the natural boundary condition.


Note

For the **Magnetic and Electric Fields** interface, a default **Electric Insulation** feature is also added to Magnetic Insulation. Also right-click to add other nodes.

Available Features

Because the These features are available for this interface and listed in alphabetical order. Also see Table 6-1 for a list of interior and exterior boundary conditions. Most features are described for the **Magnetic Fields** and **Electrostatics** interfaces.

- Ampère's Law
- Ampère's Law and Current Conservation
- Coil Group Domain
- Edge Current
- Electric Insulation
- Electric Point Dipole
- Electric Point Dipole (on Axis)
- Electric Potential
- External Current Density
- Floating Potential
- Force Calculation
- Gauge Fixing for A-field
- Ground
- Impedance Boundary Condition
- Initial Values
- Lumped Port
- Magnetic Field
- Magnetic Insulation (the default boundary condition)

- Magnetic Point Dipole
- Magnetic Potential
- Multi-Turn Coil Domain
- Normal Current Density
- Perfect Magnetic Conductor
- Periodic Condition
- Single-Turn Coil Domain
- Surface Current
- Terminal
- Thin Low Permeability Gap
- Transition Boundary Condition
- Velocity (Lorentz Term)


See Also

- Sector Symmetry

In the *COMSOL Multiphysics User's Guide*:

- About Infinite Element Domains and Perfectly Matched Layers
- Continuity on Interior Boundaries
- Identity and Contact Pairs
- Specifying Boundary Conditions for Identity Pairs
- Destination Selection


Important

The links to the features described in the *COMSOL Multiphysics User's Guide* do not work in the PDF, only from within the online help.


Tip

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

Table 6-1 lists the interior and exterior boundaries available with this interface.

TABLE 6-1: INTERIOR AND EXTERIOR ELECTRIC AND MAGNETIC BOUNDARY CONDITIONS FOR THE MAGNETIC AND ELECTRIC FIELDS INTERFACE

FEATURE	INTERIOR	EXTERIOR
MAGNETIC BOUNDARIES		
Impedance Boundary Condition		x
Lumped Port	x	x
Magnetic Field		x
Magnetic Insulation	x	x
Magnetic Potential	x	x
Perfect Magnetic Conductor	x	x
Surface Current	x	x
Thin Low Permeability Gap	x	
Transition Boundary Condition	x	
ELECTRIC BOUNDARIES		
Electric Insulation	x	x
Electric Potential	x	x
Floating Potential	x	x
Ground	x	x
Normal Current Density		x
Periodic Condition		x
Terminal	x	x


1D Axi


2D Axi

For axisymmetric models, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an **Axial Symmetry** feature to the model that is valid on the axial symmetry boundaries only.

Ampère's Law and Current Conservation

The **Ampère's Law and Current Conservation** node adds Ampère's law and the equation of continuity for the electric current. It provides an interface for defining the constitutive relations and their associated properties.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the magnetic vector potential and the equation based on Ampère's law that defines the potential.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty. If a linear temperature relation is added for the conductivity, then define the source for the temperature T. From the **Temperature** list, select an existing temperature variable (from another physics interface) if available, or select **User defined** to define a value or expression for the temperature (SI unit: K) in the field that appears underneath the list.

MATERIAL TYPE

Select a **Material type**—**Solid**, **Non-solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

CONDUCTION CURRENT


Note

See the settings for Conduction Current under Ampère's Law for the **Magnetic Fields** interface.

ELECTRIC FIELD


Note

See the settings for Electric Field under Charge Conservation for the **Electrostatics** interface.


Note

See the settings for Magnetic Field under Ampère's Law for the **Magnetic Fields** interface.

Initial Values

The **Initial Values** feature adds an initial value for the magnetic vector potential and electric potential that can serve as an initial value for a transient simulation or as an initial guess for a nonlinear solver.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define an initial value.

INITIAL VALUES

Enter a value or expression for the initial value of the **Magnetic vector potential** A (SI unit: Wb/m) and initial value of the **Electric potential** V (SI unit: V). The default values are 0.

Theory for the Magnetic and Electric Fields Interface

The Magnetic and Electric Fields Interface can be used for modeling full coupling between electric and magnetic fields. Use this physics interface for 3D, 2D in-plane, and 2D axisymmetric models. For a deeper theoretical background to the magnetic vector potential and electric scalar potential used below, see the Theory of Magnetic and Electric Fields.


Note

The Magnetic and Electric Currents interface only supports the stationary and frequency domain study types—that is, there is no transient formulation available.

In this section:

- Magnetostatics Equations
- Frequency Domain Equations

Magnetostatics Equations

To derive the magnetostatics equations, start with Ampère's law for static cases: $\nabla \times \mathbf{H} = \mathbf{J}$. Define the potentials,

$$\mathbf{B} = \nabla \times \mathbf{A}$$
$$\mathbf{E} = -\nabla V$$

and the current

$$\mathbf{J} = \sigma \mathbf{v} \times \mathbf{B} - \sigma \nabla V + \mathbf{J}^{\mathrm{e}}$$

where $\mathbf{J}^{\mathrm{e}}$ is an externally generated current density, and $\mathbf{v}$ is the velocity of the conductor.

Add the constitutive relationship, $\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$, and rewrite Ampère's law as

$$\nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + \sigma \nabla V = \mathbf{J}^{\mathrm{e}}$$

The equation of continuity is obtained by taking the divergence of Ampère's law. It is the equation solved for the electric potential. Thus the following equations for V and $\mathbf{A}$ apply:

$$-\nabla \cdot (-\sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + \sigma \nabla V - \mathbf{J}^{\mathrm{e}}) = 0$$
$$\nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + \sigma \nabla V = \mathbf{J}^{\mathrm{e}}$$

Frequency Domain Equations

To derive the time harmonic equation this physics interface solves, start with Maxwell-Ampère's law including displacement current (then called Maxwell-Ampère's law). Including this does not involve any extra computational cost in the frequency domain. Assume time-harmonic fields

$$\nabla \times \mathbf{H} = \mathbf{J} = \sigma(\mathbf{E} + \mathbf{v} \times \mathbf{B}) + j\omega \mathbf{D} + \mathbf{J}^{\mathrm{e}}$$

and use the definitions of the fields,

$$\mathbf{B} = \nabla \times \mathbf{A}$$
$$\mathbf{E} = -\nabla V - j\omega \mathbf{A}$$

and combine them with the constitutive relationships $\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$ and $\mathbf{D} = \varepsilon_0 \mathbf{E} + \mathbf{P}$ to rewrite Ampère's law as

$$(j\omega\sigma - \omega^2 \varepsilon_0)\mathbf{A} + \nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + (\sigma + j\omega\varepsilon_0)\nabla V - j\omega \mathbf{P} = \mathbf{J}^{\mathrm{e}}$$

The equation of continuity is again obtained by taking the divergence of Ampère's law. It is the equation solved for the electric potential. Thus the following equations for V and $\mathbf{A}$ apply:

$$-\nabla \cdot ((j\omega\sigma - \omega^2 \varepsilon_0)\mathbf{A} - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + (\sigma + j\omega\varepsilon_0)\nabla V - (\mathbf{J}^{\mathrm{e}} + j\omega \mathbf{P})) = 0$$
$$(j\omega\sigma - \omega^2 \varepsilon_0)\mathbf{A} + \nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + (\sigma + j\omega\varepsilon_0)\nabla V - j\omega \mathbf{P} = \mathbf{J}^{\mathrm{e}}$$

7

The Electrical Circuit Interface

The Electrical Circuit interface, found under the **AC/DC** branch ( ) in the **Model Wizard**, has the equations for modeling electrical circuits with or without connections to a distributed fields model, solving for the voltages, currents and charges associated with the circuit elements.

In this chapter:

The Electrical Circuit Interface

The **Electrical Circuit** interface ( ), found under the **AC/DC** branch ( ) in the **Model Wizard**, has the equations for modeling electrical circuits with or without connections to a distributed fields model, solving for the voltages, currents and charges associated with the circuit elements.

When this interface is added, it adds a default **Ground Node** feature and associates that with node zero in the electrical circuit.

Important	Circuit nodes are nodes in the electrical circuit and should not be confused with nodes in the model tree of COMSOL Multiphysics. Circuit node names are not restricted to numerical values but can be arbitrary character strings.

Model	Inductor in an Amplifier Circuit: Model Library path **ACDC_Module/Inductive_Devices_and_Coils/inductor_in_circuit**

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `cir`.


See Also

- Theory for the Electrical Circuit Interface
- Connecting to Electrical Circuits
- Ground Node
- Resistor
- Capacitor
- Inductor
- Voltage Source
- Current Source
- Voltage-Controlled Voltage Source
- Voltage-Controlled Current Source
- Current-Controlled Voltage Source
- Current-Controlled Current Source
- Subcircuit Definition
- Subcircuit Instance
- NPN BJT
- n-Channel MOSFET
- Diode
- External I vs. U
- External U vs. I
- External I-Terminal
- SPICE Circuit Import

Ground Node

The **Ground Node** node ( ) adds a ground node with the default node number zero to the electrical circuit. This is the default feature in the **Electrical Circuit** interface.

GROUND CONNECTION

Set the **Node name** for the ground node in the circuit. The convention is to use zero for the ground node.

Resistor

The **Resistor** node ( ) connects a resistor between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the resistor. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the **Resistance** of the resistor.

Capacitor

The **Capacitor** node ( ) connects a capacitor between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the capacitor. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the **Capacitance** of the capacitor.

Inductor

The **Inductor** node ( ) connects an inductor between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the inductor. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the **Inductance** of the inductor.

Voltage Source

The **Voltage Source** node ( ) connects a voltage source between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the voltage source. The first node represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the **Source type** that should be adapted to the selected study type. It can be **DC-source**, **AC-source**, or a time-dependent **Sine source**. Depending on the choice of source, also specify the **Voltage**, $\mathbf{V_{src}}$, the offset **Voltage**, $\mathbf{V_{off}}$, the **Frequency**, and the **Source phase**. All values are peak values rather than RMS.


Note

For the AC source, the frequency is a global input set by the solver so do not use the **Sine source** unless the model is time dependent.

Current Source

The **Current Source** node () connects a current source between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the current source. The first node represents the positive reference terminal from which the current flows through the source to the second node. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the **Source type** which should be adapted to the selected study type. It can be **DC-source**, **AC-source** or a time-dependent **Sine source**. Depending on the choice of source, also specify the **Current**, $\mathbf{I_{src}}$, the offset **Current**, $\mathbf{I_{off}}$, the **Frequency** and the **Source phase**. All values are peak values rather than RMS.


Note

For the AC source, the frequency is a global input set by the solver so do not use the **Sine source** unless the model is time-dependent.

Voltage-Controlled Voltage Source

The **Voltage-Controlled Voltage Source** node ( ) connects a voltage-controlled voltage source between two nodes in the electrical circuit. A second pair of nodes define the input control voltage.

NODE CONNECTIONS

Specify four **Node names**: the first pair for the connection nodes for the voltage source and the second pair defining the input control voltage. The first node in a pair represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the voltage **Gain**. The resulting voltage is this number multiplied by the control voltage.

Voltage-Controlled Current Source

The **Voltage-Controlled Current Source** node ( ) connects a voltage-controlled current source between two nodes in the electrical circuit. A second pair of nodes define the input control voltage.

NODE CONNECTIONS

Specify four **Node names**: the first pair for the connection nodes for the current source and the second pair defining the input control voltage. The first node in a pair represents the positive voltage reference terminal or the one from which the current flows through the source to the second node. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the voltage **Gain**. The resulting current is this number multiplied by the control voltage. Thus it formally has the unit of conductance.

Current-Controlled Voltage Source

The **Current-Controlled Voltage Source** node ( ) connects a current-controlled voltage source between two nodes in the electrical circuit. The input control current is the one flowing through a named device that must be a two-pin device.

NODE CONNECTIONS

Set two **Node names** for the connection nodes for the voltage source. The first node in a pair represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the voltage **Gain** and the **Device** (any two-pin device) name. The resulting voltage is this number multiplied by the control current through the named **Device** (any two-pin device). Thus it formally has the unit of resistance.

Current-Controlled Current Source

The **Current-Controlled Current Source** node ( ) connects a current-controlled current source between two nodes in the electrical circuit. The input control current is the one flowing through a named device that must be a two-pin device.

NODE CONNECTIONS

Specify two **Node names** for the connection nodes for the current source. The first node in a pair represents the positive reference terminal from which the current flows through the source to the second node. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the current **Gain** and the **Device** (any two-pin-device) name. The resulting current is this number multiplied by the control current through the named **Device** (any two-pin device).

Subcircuit Definition

The **Subcircuit Definition** node ( ) is used to define subcircuits. Right-click a subcircuit definition node to add all circuit features available except for the subcircuit definition feature itself. Also right-click to **Rename** the node.

SUBCIRCUIT PINS

Define the **Pin names** at which the subcircuit connects to the main circuit or to other subcircuits when referenced by a Subcircuit Instance feature. The **Pin names** refer to circuit nodes in the subcircuit. The order in which the **Pin names** are defined is the order in which they are referenced by a Subcircuit Instance feature.

Subcircuit Instance

The **Subcircuit Instance** node ( ) is used to refer to defined subcircuits.

NODE CONNECTIONS

Select the **Name of subcircuit link** from the list of defined subcircuits in the circuit model and the circuit **Node names** at which the subcircuit instance connects to the main circuit or to another subcircuit if used therein.

NPN BJT

The **NPN BJT** device model ( ) is a large-signal model for an NPN bipolar junction transistor (BJT). It is an advanced device model and no thorough description and motivation of the many input parameters is attempted here. The interested reader is referred to Ref. 2 for more details on semiconductor modeling within circuits. Many device manufacturers provide model input parameters for this BJT model. For any particular make of BJT, the device manufacturer should be the primary source of information.

NODE CONNECTIONS

Specify three **Node names** for the connection nodes for the **NPN BJT** device. These represent the *collector*, *base*, and *emitter* nodes, respectively. If the ground node is involved, the convention is to use zero for this.

MODEL PARAMETERS

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular BJT, the device manufacturer should be the primary source of information.

See Also

For an explanation of the **Model Parameters** see NPN Bipolar Transistor.

n-Channel MOSFET

The **n-Channel MOSFET** device model ( ) is a large-signal model for an n-Channel MOS transistor (MOSFET). It is an advanced device model and no thorough description and motivation of the many input parameters is attempted here. The interested reader is referred to Ref. 2 for more details on semiconductor modeling

within circuits. Many device manufacturers provide model parameters for this MOSFET model. For any particular make of MOSFET, the device manufacturer should be the primary source of information.

NODE CONNECTIONS

Specify four **Node names** for the connection nodes for the **n-Channel MOSFET** device. These represent the *drain*, *gate*, *source*, and *bulk* nodes, respectively. If the ground node is involved, the convention is to use zero for this.

MODEL PARAMETERS

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular MOSFET, the device manufacturer should be the primary source of information.

For an explanation of the **Model Parameters** see n-Channel MOS Transistor.

Diode

The **Diode** device model ( ) is a large-signal model for a diode. It is an advanced device model and no thorough description and motivation of the many input parameters is attempted here. The interested reader is referred to Ref. 2 for more details on semiconductor modeling within circuits. Many device manufacturers provide model parameters for this diode model. For any particular make of diode, the device manufacturer should be the primary source of information.

NODE CONNECTIONS

Specify two **Node names** for the positive and negative nodes for the **Diode** device. If the ground node is involved, the convention is to use zero for this.

MODEL PARAMETERS

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular diode, the device manufacturer should be the primary source of information.

For an explanation of the **Model Parameters** see Diode.

External I vs. U

The **External I vs. U** node ( ) connects an arbitrary voltage measurement, for example a circuit terminal or circuit port boundary or a coil domain from another physics interface, as a source between two nodes in the electrical circuit. The resulting circuit current from the first node to the second node is typically coupled back as a prescribed current source in the context of the voltage measurement.

NODE CONNECTIONS

Specify the two **Node names** for the connecting nodes for the voltage source. The first node represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this.

EXTERNAL DEVICE

Enter the source of the **Voltage**. If circuit or current excited terminals or circuit ports are defined on boundaries or a multiturn coil domains is defined in other physics interfaces, these display as options in the **Voltage** list. Also select the **User defined** option and enter your own voltage variable, for example, using a suitable coupling operator. For inductive or electromagnetic wave propagation models, the voltage measurement must be performed as an integral of the electric field as the electric potential only does not capture induced EMF. Also the integration must be performed over a distance that is short compared to the local wavelength.

Important	Except for when coupling to a circuit terminal or circuit port, the current flow variable must be manually coupled back in the electrical circuit to the context of the voltage measurement. This applies also when coupling to a current excited terminal. The name of this current variable follows the convention `cirn.IvsUm_i`, where `cirn` is the tag of the Electrical Circuit interface node and `IvsUm` is the tag of the **External I vs. U** node. The mentioned tags are typically displayed within curly braces `{}` in the model tree.

See Also	Model Couplings in the *COMSOL Multiphysics User's Guide*

External U vs. I

The **External U vs. I** node ( ) connects an arbitrary current measurement, for example, from another physics interface, as a source between two nodes in the electrical circuit. The resulting circuit voltage between the first node and the second node is typically coupled back as a prescribed voltage source in the context of the current measurement.

NODE CONNECTIONS

Specify the two **Node names** for the connecting nodes for the current source. The current flows from the first node to the second node. If the ground node is involved, the convention is to use zero for this.

EXTERNAL DEVICE

Enter the source of the **Current**. Voltage excited terminals or lumped ports defined on boundaries in other physics interfaces are natural candidates but do not appear as options in the **Voltage** list because those do not have an accurate built-in current measurement variable. A **User defined** option must be selected and a current variable entered, for example, using a suitable coupling operator.

Important	The voltage variable must be manually coupled back in the electrical circuit to the context of the current measurement. This applies also when coupling to a voltage excited terminal or lumped port. The name of this voltage variable follows the convention `cirn.UvsIm_v`, where `cirn` is the tag of the Electrical Circuit interface node and `UvsIm` is the tag of the **External U vs. I** node. The mentioned tags are typically displayed within curly braces `{}` in the model tree.

See Also	Model Couplings in the *COMSOL Multiphysics User's Guide*

External I-Terminal

The **External I-Terminal** node ( ) connects an arbitrary voltage-to-ground measurement, for example, a circuit terminal boundary from another physics interface, as a voltage-to-ground assignment to a node in the electrical circuit. The resulting

circuit current from the node is typically coupled back as a prescribed current source in the context of the voltage measurement. This feature does not apply when coupling to inductive or electromagnetic wave propagation models as then voltage must be defined as a line integral between two points rather than a single point measurement of electric potential. For such couplings, use the **External I vs. U** feature instead.

NODE CONNECTIONS

Set the **Node name** for the connecting node for the voltage assignment.

EXTERNAL TERMINAL

Enter the source of the **Voltage**. If circuit- or current-excited terminals are defined on boundaries in other physics interfaces, these display as options in the **Voltage** list. Also select the **User defined** option and enter a voltage variable, for example, using a suitable coupling operator.


Important

Except for when coupling to a circuit terminal, the current flow variable must be manually coupled back in the electrical circuit to the context of the voltage measurement. This applies also when coupling to a current excited terminal. The name of this current variable follows the convention `cirn.termIm_i`, where `cirn` is the tag of the Electrical Circuit interface node and `termIm` is the tag of the **External I-Terminal** node. The mentioned tags are typically displayed within curly braces `{}` in the model tree.


See Also

Model Couplings in the *COMSOL Multiphysics User's Guide*

SPICE Circuit Import

Right-click the **Electrical Circuit** node ( ) to import an existing SPICE netlist (select **Import Spice Netlist**). A window opens—enter a file location or browse your directories to find one. The default file extension for a SPICE netlist is `.cir`. The SPICE circuit import translates the imported netlist into Electrical Circuit interface nodes so these define the subset of SPICE features that can be imported.

Theory for the Electrical Circuit Interface

The Electrical Circuit Interface theory is discussed in this section:

- Electric Circuit Modeling and the Semiconductor Device Models
- NPN Bipolar Transistor
- n-Channel MOS Transistor
- Diode
- References for the Electrical Circuit Interface

Electric Circuit Modeling and the Semiconductor Device Models

Electrical circuit modeling capabilities are useful when simulating all sorts of electrical and electromechanical devices ranging from heaters and motors to advanced plasma reactors in the semiconductor industry. There are two fundamental ways that an electrical circuit model relates to a physical field model.

- Either the field model is used to get a better, more accurate description of a single device in the electrical circuit model or
- the electrical circuit is used to drive or terminate the device in the field model in such a way that it makes more sense to simulate both as a tightly coupled system.

The Electrical Circuit interface makes it is possible to add nodes representing circuit elements directly to the model tree in a COMSOL Multiphysics model. The circuit variables can then be connected to a physical device model to perform co-simulations of circuits and multiphysics. The model acts as a device connected to the circuit so that its behavior is analyzed in larger systems.

The fundamental equations solved by the electrical circuit interface are Kirchhoff's circuit laws, which in turn can be deduced from Maxwell's equations. The supported study types are Stationary, Frequency Domain, and Time Dependent.


Note

The circuit definition in COMSOL Multiphysics adheres to the SPICE format developed at University of California, Berkeley (Ref. 1) and SPICE netlists can also be imported, generating the corresponding features in the COMSOL Multiphysics model. Most circuit simulators can export to this format or some dialect of it.

There are three more advanced large-signal semiconductor device features available in the Electrical Circuit interface. The equivalent circuits and the equations defining their non-ideal circuit elements are described in this section. For a more detailed account on semiconductor device modeling, see Ref. 2.

NPN Bipolar Transistor

Figure 7-1 illustrates the equivalent circuit for the bipolar transistor.

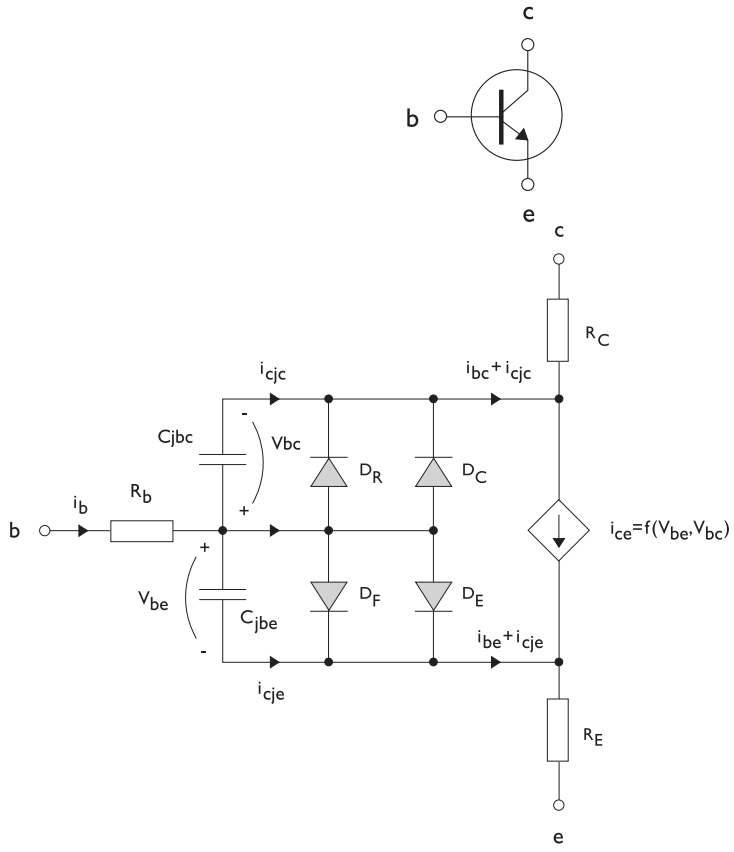


Figure 7-1: A circuit for the bipolar transistor.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$v_{rb} = \frac{1}{A}\left(R_{BM} - \frac{R_B - R_{BM}}{f_{bq}}\right) i_b$$

$$f_{bq} = \frac{1}{2\left(1 - \frac{v_{bc}}{V_{AF}} - \frac{v_{be}}{V_{AR}}\right)}\left(1 + \sqrt{1 + 4I_S\left(\frac{e^{-\frac{v_{be}}{N_F V_T}} - 1}{I_{KF}A} + \frac{e^{-\frac{v_{bc}}{N_R V_T}} - 1}{I_{KR}A}\right)}\right)$$

$$i_{be} = A\left(\frac{I_S}{B_F}\left(e^{-\frac{v_{be}}{N_F V_T}} - 1\right) + I_{SE}\left(e^{-\frac{v_{be}}{N_E V_T}} - 1\right)\right)$$

$$i_{bc} = A\left(\frac{I_S}{B_R}\left(e^{-\frac{v_{bc}}{N_R V_T}} - 1\right) + I_{SC}\left(e^{-\frac{v_{bc}}{N_C V_T}} - 1\right)\right)$$

$$i_{ce} = A\left(\frac{I_S}{f_{bq}}\left(e^{-\frac{v_{be}}{N_F V_T}} + e^{-\frac{v_{bc}}{N_C V_T}}\right)\right)$$

$$V_T = \frac{k_B T_{NOM}}{q}$$

There are also two capacitances that use the same formula as the junction capacitance of the diode model. In the parameter names below, replace x with C for the base-collector capacitance and E for the base-emitter capacitance.

$$C_{jbx} = AC_{Jx} \times \begin{cases} \left(1 - \frac{v_{bx}}{V_{Jx}}\right)^{-M_{Jx}} & v_{bx} < F_C V_{Jx} \\ (1 - F_C)^{-1 - M_{Jx}}\left(1 - F_C(1 + M_{Jx}) + M_{Jx}\frac{v_{bx}}{V_{Jx}}\right) & v_{bx} \geq F_C V_{Jx} \end{cases}$$

The model parameters are listed in the table below.

TABLE 7-1: BIPOLAR TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
B_F	100	Ideal forward current gain
B_R	1	Ideal reverse current gain
C_{JC}	0 F/m^2	Base-collector zero-bias depletion capacitance
C_{JE}	0 F/m^2	Base-emitter zero-bias depletion capacitance
F_C	0.5	Breakdown current
I_{KF}	Inf (A/m^2)	Corner for forward high-current roll-off
I_{KR}	Inf (A/m^2)	Corner for reverse high-current roll-off

TABLE 7-1: BIPOLAR TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
I_S	1e-15 A/m^2	Saturation current
I_{SC}	0 A/m^2	Base-collector leakage saturation current
I_{SE}	0 A/m^2	Base-emitter leakage saturation current
M_{JC}	1/3	Base-collector grading coefficient
M_{JE}	1/3	Base-emitter grading coefficient
N_C	2	Base-collector ideality factor
N_E	1.4	Base-emitter ideality factor
N_F	1	Forward ideality factor
N_R	1	Reverse ideality factor
R_B	0 Ωm^2	Base resistance
R_{BM}	0 Ωm^2	Minimum base resistance
R_C	0 Ωm^2	Collector resistance
R_E	0 Ωm^2	Emitter resistance
T_{NOM}	298.15 K	Device temperature
V_{AF}	Inf (V)	Forward Early voltage
V_{AR}	Inf (V)	Reverse Early voltage
V_{JC}	0.71 V	Base-collector built-in potential
V_{JE}	0.71 V	Base-emitter built-in potential

n-Channel MOS Transistor

Figure 7-2 illustrates an equivalent circuit for the MOS transistor.

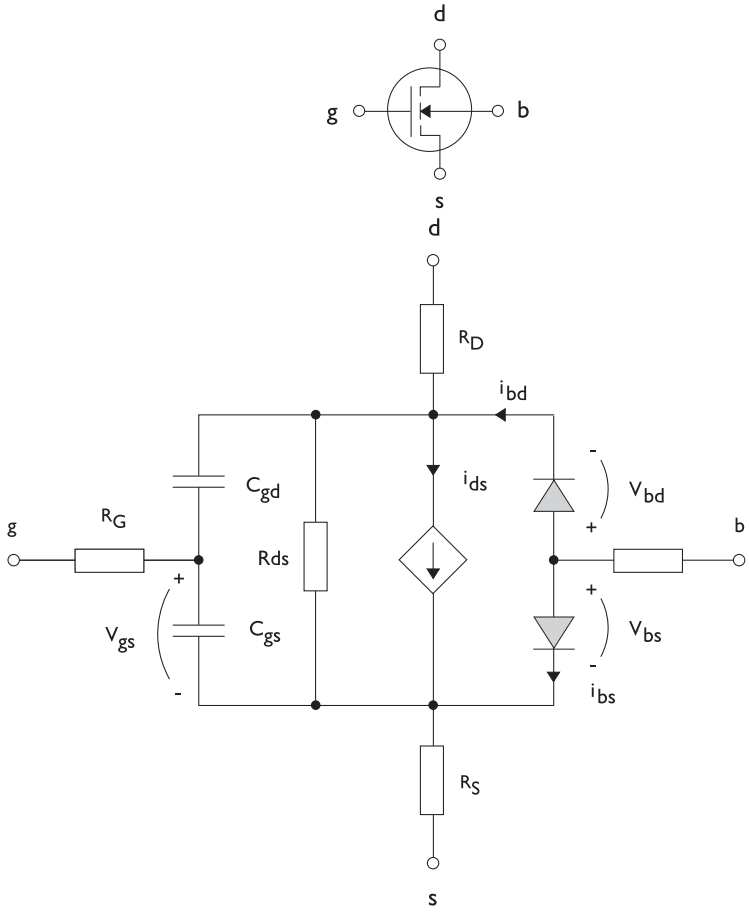


Figure 7-2: A circuit for the MOS transistor.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$i_{ds} = \begin{cases} \frac{W}{L}\frac{K_P}{2}(1+\Lambda v_{ds})v_{ds}(2v_{th}-v_{ds}) & v_{ds} < v_{th} \\ \frac{W}{L}\frac{K_P}{2}(1+\Lambda v_{ds})v_{th}^2 & v_{ds} \geq v_{th} \\ 0 & v_{ds} < v_{th} \leq 0 \end{cases}$$

$$v_{th} = v_{gs} - (V_{TO} + \Gamma(\sqrt{\Phi - v_{bs}} - \sqrt{\Phi}))$$

$$i_{bd} = I_S\left(e^{-\frac{v_{bd}}{NV_T}} - 1\right)$$

$$i_{bs} = I_S\left(e^{-\frac{v_{bs}}{NV_T}} - 1\right)$$

$$V_T = \frac{k_B T_{NOM}}{q}$$

There are also several capacitances between the terminals

$$C_{gd} = C_{gd0}W$$

$$C_{gs} = C_{gs0}W$$

$$C_{jbd} = C_{BD} \times \begin{cases} \left(1 - \frac{v_{bd}}{P_B}\right)^{-M_J} & v_{bx} < F_C P_B \\ (1-F_C)^{-1-M_J}\left(1 - F_C(1+M_J) + M_J\frac{v_{bx}}{P_B}\right) & v_{bx} \geq F_C P_B \end{cases}$$

The model parameters are as follows:

TABLE 7-2: MOS TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
C_{BD}	0 F/m	Bulk-drain zero-bias capacitance
C_{GDO}	0 F/m	Gate-drain overlap capacitance
C_{GSO}	0 F/m	Gate-source overlap capacitance
F_C	0.5	Capacitance factor
I_S	1e-13 A	Bulk junction saturation current
K_P	2e-5 A/V^2	Transconductance parameter
L	50e-6 m	Gate length
M_J	0.5	Bulk junction grading coefficient
N	1	Bulk junction ideality factor

TABLE 7-2: MOS TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
P_B	0.75 V	Bulk junction potential
R_B	0 Ω	Bulk resistance
R_D	0 Ω	Drain resistance
R_{DS}	Inf (Ω)	Drain-source resistance
R_G	0 Ω	Gate resistance
R_S	0 Ω	Source resistance
T_{NOM}	298.15 K	Device temperature
V_{TO}	0 V	Zero-bias threshold voltage
W	50e-6 m	Gate width
Γ (GAMMA)	0 $V^{0.5}$	Bulk threshold parameter
Φ (PHI)	0.5 V	Surface potential
Λ (LAMBDA)	0 1/V	Channel-length modulation

Diode

Figure 7-3 illustrates equivalent circuit for the diode.

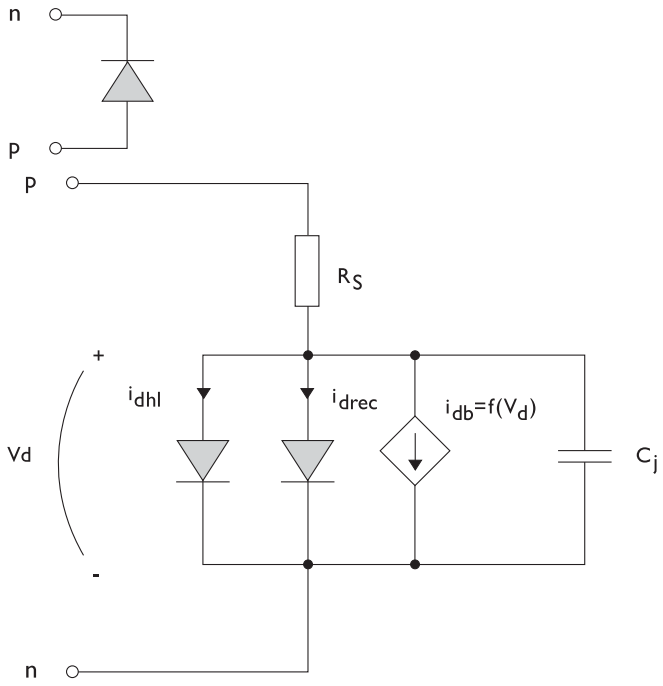


Figure 7-3: A circuit for the diode.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$i_d = i_{dhl} + i_{drec} + i_{db} + i_c$$

$$i_{dhl} = I_S\left(e^{-\frac{v_d}{NV_T}} - 1\right)\frac{1}{\sqrt{1 + \frac{I_S}{I_{KF}}\left(e^{-\frac{v_d}{NV_T}} - 1\right)}}$$

$$i_{drec} = I_{SR}\left(e^{-\frac{v_d}{N_R V_T}} - 1\right)$$

$$i_{db} = I_{BV}e^{-\frac{v_d + B_V}{N_{BV}V_T}}$$

$$C_j = C_{J0} \times \begin{cases} \left(1 - \frac{v_d}{V_J}\right)^{-M} & v_d < F_C V_J \\ (1 - F_C)^{-1-M}\left(1 - F_C(1+M) + M\frac{v_d}{V_J}\right) & v_d \geq F_C V_J \end{cases}$$

$$V_T = \frac{k_B T_{NOM}}{q}$$

where the following model parameters are required

TABLE 7-3: DIODE TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
B_V	Inf (V)	Reverse breakdown voltage
C_{J0}	0 F	Zero-bias junction capacitance
F_C	0.5	Forward-bias capacitance coefficient
I_{BV}	1e-09 A	Current at breakdown voltage
I_{KF}	Inf (A)	Corner for high-current roll-off
I_S	1e-13 A	Saturation current
M	0.5	Grading coefficient
N	1	Ideality factor
N_{BV}	1	Breakdown ideality factor
N_R	2	Recombination ideality factor
R_S	0 Ω	Series resistance
T_{NOM}	298.15 K	Device temperature
V_J	1.0 V	Junction potential

References for the Electrical Circuit Interface

1. http://bwrc.eecs.berkeley.edu/Classes/IcBook/SPICE/

2. P. Antognetti and G. Massobrio, *Semiconductor Device Modeling with Spice*, 2nd ed., McGraw-Hill, Inc., 1993.

8

The Heat Transfer Branch

The AC/DC Module license includes an interface found under the **Heat Transfer>Electromagnetic Heating** branch ( ) in the **Model Wizard**. This interface combines magnetic fields with heat transfer for modeling of electromagnetic heating such as induction heating.

In this chapter:

The Induction Heating Interface

The **Induction Heating** interface ( ), found under the **Heat Transfer>Electromagnetic Heating** branch ( ) in the **Model Wizard**, combines all features from the Magnetic Fields interface in the time-harmonic formulation with the Heat Transfer interface for modeling of induction heating and eddy current heating.

The interface has the equations, boundary conditions, and sources for modeling such electromagnetic 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). Thus, it is associated with two predefined study types:

- **Frequency-Stationary**—time-harmonic magnetic fields and stationary heat transfer
- **Frequency-Transient**—time-harmonic magnetic fields and transient heat transfer.

When this interface is added, the following default nodes are also added to the **Model Builder**— **Induction Heating Model**, **Electromagnetic Heat Source**, **Boundary Electromagnetic Heat Source**, **Thermal Insulation**, **Magnetic Insulation**, and **Initial Values**. Right-click the **Induction Heating** node to add other features that implement, for example, boundary conditions and sources.


Model

Inductive Heating of a Copper Cylinder: Model Library path **ACDC_Module/Electromagnetic_Heating/inductive_heating**

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `ih`.

DOMAIN SELECTION

The default setting is to include **All domains** in the model. To choose specific domains, select **Manual** from the **Selection** list.

PHYSICAL MODEL

Select the **Out-of-plane heat transfer** check box (2D models only) to include heat transfer out of the plane.

If your license includes the Heat Transfer Module, you can select the **Surface-to-surface radiation** check box to include surface-to-surface radiation as part of the heat transfer. This adds a **Radiation Settings** section.See the **Physical Model** section in The Heat Transfer Interface for details.

If your license includes the Heat Transfer Module, you can select the **Radiation in participating media** check box to include radiation in participating media as part of the heat transfer. This adds a **Participating media Settings** section.See the **Physical Model** section in The Heat Transfer Interface for details.

If your license includes the Heat Transfer Module, you can select the **Heat Transfer in biological tissue** check box to enable the **Biological Tissue** feature. See the **Physical Model** section in The Bioheat Transfer Interface for details.

CONSISTENT AND INCONSISTENT STABILIZATION

To display this section, click the **Show** button ( ) and select **Stabilization**. There are two consistent stabilization methods available—**Streamline diffusion** and **Crosswind diffusion**. Streamline diffusion is active by default. There is one inconsistent stabilization method, **Isotropic diffusion**, which is not activated by default.

BACKGROUND FIELD

Select an option from the **Solve for** list—**Reduced field** or **Full field** (the default). If **Reduced field** is selected, specify a **Background magnetic vector potential** $\mathbf{A}_b$ (SI unit: Wb/m). The total field used in the physics and equations are given by the sum of the reduced and background fields.

COMPONENTS

 *2D*  *2D Axi*	The **Components** section is only available in 2D and 2D axially symmetric models. Select **Components**—**Out-of-plane vector** (the default), **In-plane vector**, or **Three-component vector** for the magnetic vector potential. From the practical viewpoint this choice is equivalent to deciding in what directions the electric current is allowed to flow (out-of-plane currents, in-plane currents, or currents flowing in all three coordinate directions).

THICKNESS

 2D	Enter a value or expression for the **Out-of-plane thickness** d (SI unit: m). The default value of 1 m is typically not representative for a thin domain. Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D models.

SWEEP SETTINGS

Select the **Activate terminal sweep** check box and enter a **Sweep parameter name** in the field. The default is `PortName`.

DEPENDENT VARIABLES

The dependent variables (field variables) are for the **Temperature** T and the **Magnetic Vector potential** A. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. Select **Quadratic**, **Linear, Cubic**, or **Quartic** for the **Temperature**, **Surface radiosity**, and **Magnetic vector potential**. Specify the **Value type when using splitting of complex variables**—**Real** or **Complex** (the default).

 See Also	• The Model Builder Show and Hide Physics Options • Domain, Boundary, Edge, Point, and Pair Features for the Induction Heating Interface • Theory of Magnetic and Electric Fields • Theory for the Heat Transfer Interfaces in the *COMSOL Multiphysics User's Guide:*

Domain, Boundary, Edge, Point, and Pair Features for the Induction Heating Interface

The Induction Heating Interface shares most of its settings windows with the **Magnetic Fields**, **Heat Transfer**, and **Joule Heating** interfaces.

These domain, boundary, edge, point, and pair features are described in this guide for the **Magnetic Fields** interface (listed in alphabetical order):

- Ampère's Law
- Edge Current
- Electric Point Dipole
- Electric Point Dipole (on Axis)
- Electromagnetic Heat Source
- External Current Density
- Force Calculation
- Gauge Fixing for A-field
- Induction Heating Model
- Infinite Elements
- Initial Values
- Impedance Boundary Condition
- Lumped Port
- Magnetic Field
- Magnetic Insulation
- Magnetic Point Dipole
- Magnetic Potential
- Periodic Condition
- Perfect Magnetic Conductor
- Sector Symmetry
- Surface Current
- Thin Low Permeability Gap
- Transition Boundary Condition

These domain, boundary, edge, point, and pair features are described for the **Heat Transfer** and **Joule Heating** interfaces in the *COMSOL Multiphysics User's Guide* (listed in alphabetical order):


Important

The links to features described the *COMSOL Multiphysics User's Guide* do not work in the PDF, only from within the online help.


Tip

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

- Auxiliary Dependent Variable
- Boundary Electromagnetic Heat Source
- Boundary Heat Source and Pair Boundary Heat Source
- Continuity on Interior Boundaries
- Heat Flux
- Heat Source
- Heat Transfer in Fluids
- Heat Transfer in Solids
- Line Heat Source
- Outflow
- Point Heat Source
- The Pointwise Constraint Node
- Surface-to-Ambient Radiation
- Symmetry
- Temperature
- Thermal Insulation
- Thin Thermally Resistive Layer and Pair Thin Thermally Resistive Layer
- Translational Motion
- The Weak Constraint Node
- The Weak Contribution Node

- The Weak Contributions on Mesh Boundaries Node

Induction Heating Model

The **Induction Heating Model** feature has settings to define the **Conduction Current**, **Electric Field**, **Magnetic Field**, **Heat Conduction**, and **Thermodynamics**.

DOMAIN SELECTION

The default feature settings cannot be edited and include all domains in the model.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty. If a linear temperature relation is added for the conductivity, then define the source for the temperature T. From the **Temperature** list, select an existing temperature variable (from another physics interface) if available, or select **User defined** to define a value or expression for the temperature (SI unit: K) in the field that appears underneath the list.

CONDUCTION CURRENT


Note

See the settings for Conduction Current under Ampère's Law for the **Magnetic Fields** interface.

ELECTRIC FIELD


Note

See the settings for Electric Field under Charge Conservation described for the **Electrostatics** interface.

MAGNETIC FIELD


Note

See the settings for Magnetic Field under Ampère's Law for the **Magnetic Fields** interface, with one difference:

- **HB curve**. Do not select this option for time harmonic modeling as it is not relevant when using the **Induction Heating** interface.

HEAT CONDUCTION

The default **Thermal conductivity** k (SI unit: W/(m·K)) uses values **From material**. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity and enter other values or expressions in the field or matrix.

The thermal conductivity k describes the relationship between the heat flux vector $\mathbf{q}$ and the temperature gradient ∇T as in $\mathbf{q} = -k\nabla T$, which is *Fourier's law of heat conduction*. Enter this quantity as power per length and temperature.

THERMODYNAMICS

The default uses values **From material** for the **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)) and **Density** ρ (SI unit: kg/m^3). Select **User defined** to enter other values or expressions for one or both variables.

Electromagnetic Heat Source

The **Electromagnetic Heat Source** feature represents the electromagnetic losses, Q_e (SI unit: W/m^3), as a heat source in the heat transfer part of the model. It is given by

$$Q_\mathrm{e} = Q_\mathrm{rh} + Q_\mathrm{ml}$$

where the resistive losses are

$$Q_\mathrm{rh} = \frac{1}{2}\mathrm{Re}(\mathbf{J} \cdot \mathbf{E}^*)$$

and the magnetic losses are

$$Q_\mathrm{ml} = \frac{1}{2}\mathrm{Re}(i\omega\mathbf{B} \cdot \mathbf{H}^*)$$

DOMAIN SELECTION

From the **Selection** list, choose the domains to apply the model. The default feature settings cannot be edited and include all domains in the model.

Initial Values

The **Initial Values** feature adds initial values for the temperature, surface radiosity, and magnetic vector potential.

DOMAIN SELECTION

From the **Selection** list, choose the domains to apply the initial values. The default setting is to include all domains in the model.

INITIAL VALUES

Enter values or expressions for the **Temperature** T (SI unit: K), **Surface radiosity** J (SI unit: W/m^2), and **Magnetic vector potential** A (SI unit: Wb/m). The default temperature is 293.15 K (20 °C).

9

Materials

This chapter describes the materials databases included with the AC/DC Module.

- Material Library and Databases
- Using the AC/DC Material Database

Material Library and Databases

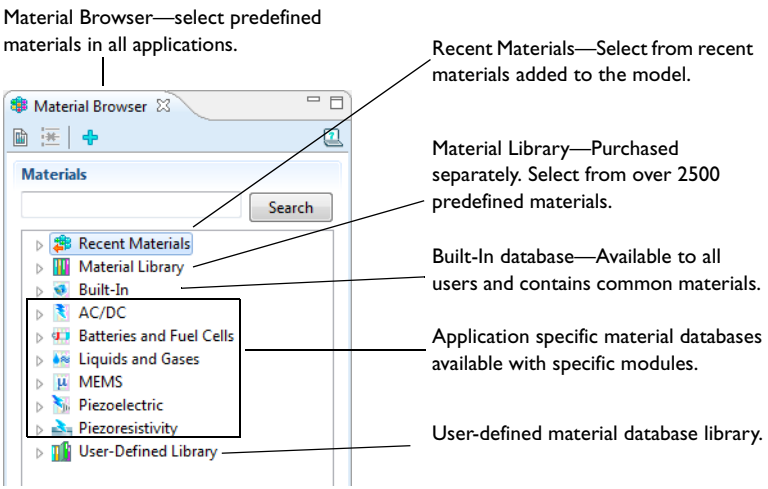
See Also	For detailed information about all the other materials databases and the separately purchased Material Library, see Materials in the *COMSOL Multiphysics User's Guide*.

Model	For an example of the AC/DC materials database, see Small-Signal Analysis of an Inductor: Model Library path **ACDC_Module/Inductive_Devices_and_Coils/small_signal_analysis_of_inductor**.

In this section:

- About the Material Databases
- About Using Materials in COMSOL
- Opening the Material Browser
- Using Material Properties

About the Material Databases



All COMSOL modules have predefined material data available to build models. The most extensive material data is contained in the separately purchased Material Library, but all modules contain commonly used or module-specific materials. For example, the *Built-In* database is available to all users but the *MEMS* database is included with the MEMS Module and Structural Mechanics Module. Also create custom materials and material libraries by researching and entering material properties.

All the material databases (including the Material Library) are accessed from the **Material Browser**. These databases are briefly described below.

RECENT MATERIALS

From the **Recent Materials** folder ( ), select from a list of recently used materials, with the most recent at the top. This folder is available after the first time a material is added to a model.

MATERIAL LIBRARY

An optional add-on database, the **Material Library** ( ), contains data for over 2500 materials and 20,000 property functions.

BUILT-IN

Included with COMSOL Multiphysics, the **Built-In** database ( ) contains common solid materials with electrical, structural, and thermal properties.

See Also

Predefined Built-In Materials for all COMSOL Modules in the *COMSOL Multiphysics User's Guide*

AC/DC

Included in the AC/DC Module, the **AC/DC** database ( ) has electric properties for some magnetic and conductive materials.

BATTERIES AND FUEL CELLS

Included in the Batteries & Fuel Cells Module, the **Batteries and Fuel Cells** database ( ) includes properties for electrolytes and electrode reactions for certain battery chemistries.

LIQUIDS AND GASES

Included in the Acoustics Module, CFD Module, Chemical Reaction Engineering Module, Heat Transfer Module, MEMS Module, Pipe Flow Module, and Subsurface Flow Module, the **Liquids and Gases** database ( ) includes transport properties and surface tension data for liquid/gas and liquid/liquid interfaces.

MEMS

Included in the MEMS Module and Structural Mechanics Module, the **MEMS** database ( ) has properties for MEMS materials—metals, semiconductors, insulators, and polymers.

PIEZOELECTRIC

Included in the Acoustics Module, MEMS Module, and Structural Mechanics Module, the **Piezoelectric** database ( ) has properties for piezoelectric materials.

PIEZORESISTIVITY

Included in the MEMS Module, the **Piezoresistivity** database ( ) has properties for piezoresistive materials, including p-Silicon and n-Silicon materials.

USER-DEFINED LIBRARY

The **User-Defined Library** folder ( ) is where user-defined materials databases (libraries) are created. When any new database is created, this also displays in the **Material Browser**.

Important	The materials databases shipped with COMSOL Multiphysics are read-only. This includes the Material Library and any materials shipped with the optional modules.

See Also	Creating Your Own User-Defined Libraries in the *COMSOL Multiphysics User's Guide*

USING THE MATERIALS IN THE PHYSICS SETTINGS

The physics set-up in a model is determined by a combination of settings in the **Materials** and physics interface nodes. When the first material is added to a model, COMSOL automatically assigns that material to the entire geometry. Different geometric entities can have different materials. The following example uses the *heat_sink.mph* model file contained in the Heat Transfer Module and CFD Module Model Libraries.

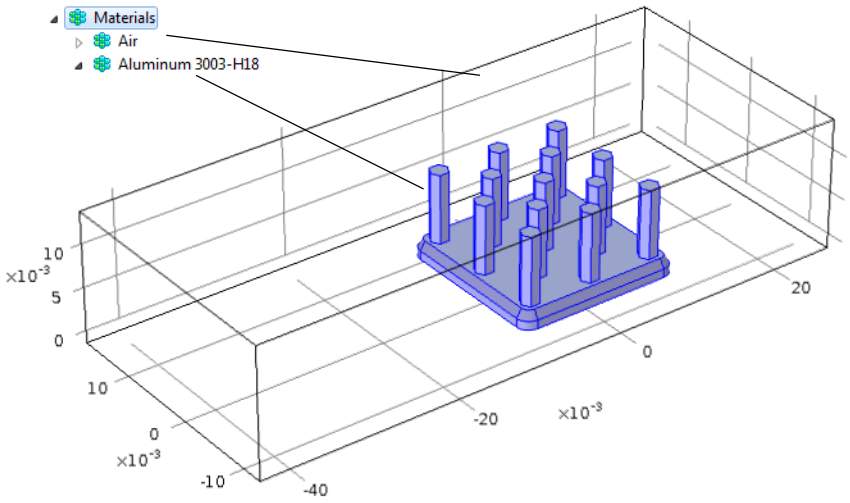


Figure 9-1: Assigning materials to a heat sink model. Air is assigned as the material to the box surrounding the heat sink, and aluminum to the heat sink itself.

If a geometry consists of a heat sink in a container, **Air** can be assigned as the material in the container surrounding the heat sink and **Aluminum** as the heat sink material itself (see Figure 9-1). The **Conjugate Heat Transfer** interface, selected during model set-up, has a **Fluid** flow model, defined in the box surrounding the heat sink, and a **Heat Transfer** model, defined in both the aluminum heat sink and in the air box. The **Heat Transfer in Solids 1** settings use the material properties associated to the **Aluminum 3003-H18** materials node, and the **Fluid 1** settings define the flow using the **Air** material properties. The other nodes under **Conjugate Heat Transfer** define the initial and boundary conditions.

All physics interface properties automatically use the correct **Materials** properties when the default **From material** setting is used. This means that one node can be used to define the physics across several domains with different materials; COMSOL then uses the material properties from the different materials to define the physics in the domains. If material properties are missing, the **Material Contents** section on the **Materials** page displays a stop icon ( ) to warn about the missing properties and a warning icon ( ) if the property exists but its value is undefined.


See Also

The Material Page in the *COMSOL Multiphysics User's Guide*

There are also some physics interface properties that by default define a material as the **Domain material** (that is, the materials defined on the same domains as the physics interface). For such material properties, select any other material that is present in the model, regardless of its selection.

EVALUATING AND PLOTTING MATERIAL PROPERTIES

You can access the material properties for evaluation and plotting like other variables in a model using the variable naming conventions and scoping mechanisms:

- To access a material property throughout the model (across several materials) and not just in a specific material, use the special material container `root.material`. For example, `root.material.rho` is the density ρ as defined by the materials in each domain in the geometry. For plotting, you can type the expression `material.rho` to create a plot that shows the density of all materials.


Note

If you use a temperature-dependent material, each material contribution asks for a special model input. For example, `rho(T)` in a material `mat1` asks for `root.mat1.def.T`, and you need to define this variable (`T`) manually—if the temperature is not available as a dependent variable—to make the density variable work.

- To access a material property from a specific material, you need to know the tags for the material and the property group. Typically, for the first material (Material 1) the tag is `mat1` and most properties reside in the default Basic property group with the tag `def`. The variable names appear in the **Variable** column in the table under **Output properties** in the settings window for the property group; for example, `Cp` for the

heat capacity at constant pressure. The syntax for referencing the heat capacity at constant pressure in Material 1 is then `mat1.def.Cp`. Some properties are anisotropic tensors, and each of the components can be accessed, such as `mat1.def.k11`, `mat1.def.k12`, and so on, for the thermal conductivity. For material properties that are functions, call these with input arguments such as `mat1.def.rho(pA,T)` where `pA` and `T` are numerical values or variables representing the absolute pressure and the temperature, respectively. The functions can be plotted directly from the function nodes' settings window by first specifying suitable ranges for the input arguments.

- Many physics interfaces also define variables for the material properties that they use. For example, `solid.rho` is the density in the Solid Mechanics interface and is equal to the density in a material when it is used in the domains where the Solid Mechanics interface is active. If you define the density in the Solid Mechanics interface using another value, `solid.rho` represents that value and not the density of the material. If you use the density from the material everywhere in the model, `solid.rho` and `material.rho` are identical.

Opening the Material Browser


Note

When using the **Material Browser**, the words *window* and *page* are interchangeable. For simplicity, the instructions refer only to the **Material Browser**.

1 Open or create a model file.

2 From the **View** menu choose **Material Browser** or right-click the **Materials** node and choose **Open Material Browser**.

The **Material Browser** opens by default in the same position as the settings window.

3 Under **Material Selection**, search or browse for materials.

- Enter a **Search** term to find a specific material by name, UNS number (Material Library materials only), or DIN number (Material Library materials only). If the

search is successful, a list of filtered databases containing that material displays under **Material Selection**.

To clear the search field and browse, delete the search term and click **Search** to reload all the databases.

- Click to open each database and browse for a specific material by class (for example, in the Material Library) or physics module (for example, MEMS Materials).

Always review the material properties to confirm they are applicable for the model. For example, **Air** provides temperature-dependent properties that are valid at pressures around 1 atm.

4 When the material is located, right-click to **Add Material to Model**.

A node with the material name is added to the **Model Builder** and the **Material** page opens.

Using Material Properties

For detailed instructions, see Adding Predefined Materials and Material Properties Reference in the *COMSOL Multiphysics User's Guide*.

Using the AC/DC Material Database

All physics interfaces in the AC/DC Module support the use of the COMSOL Multiphysics material databases. The electromagnetic material properties that can be stored in the material databases are:

- Electrical conductivity and resistivity
- Relative permittivity
- Relative permeability
- Nonlinear BH-curves
- Refractive index

The AC/DC database is included with this module and contains electromagnetic and other material properties for these materials:

PREDEFINED MATERIALS
Copper
Soft Iron (without losses)
Soft Iron (with losses)
Quartz
Graphite
Graphite felt
Silicon Carbide

Some properties depend on the magnetic flux density, location, or temperature. The database contains, depending on the material and in addition to the more common material properties, the following properties:

PREDEFINED PROPERTIES
Remnant flux density
Reference temperature
Temperature coefficient
Nonlinear BH-curves
Resistivity at reference temperature

10

Glossary

This Glossary of Terms contains finite element modeling terms in an electromagnetics context. For mathematical terms as well as geometry and CAD terms specific to the COMSOL Multiphysics software and documentation, please see the glossary in the *COMSOL Multiphysics User's Guide*. For references to more information about a term, see the index.

Glossary of Terms

anisotropy Variation of material properties with direction.

constitutive relation The relation between the $\mathbf{D}$ and $\mathbf{E}$ fields and between the $\mathbf{B}$ and $\mathbf{H}$ fields. These relations depend on the material properties.

eddy currents Induced currents normal to a time-varying magnetic flux in a ferromagnetic material.

edge element See *vector element*.

electric dipole Two equal and opposite charges $+q$ and $-q$ separated a short distance d. The electric dipole moment is given by $\mathbf{p} = q\mathbf{d}$, where $\mathbf{d}$ is a vector going from $-q$ to $+q$.

gauge transformation A variable transformation of the electric and magnetic potentials that leaves Maxwell's equations invariant.

magnetic dipole A small circular loop carrying a current. The magnetic dipole moment is $\mathbf{m} = IA\mathbf{e}$, where I is the current carried by the loop, A its area, and $\mathbf{e}$ a unit vector along the central axis of the loop.

Nedelec's edge element See *vector element*.

phasor A complex function of space representing a sinusoidally varying quantity.

quasi-static approximation The electromagnetic fields are assumed to vary slowly, so that the retardation effects can be neglected. This approximation is valid when the geometry under study is considerably smaller than the wavelength.

vector element A finite element often used for electromagnetic vector fields. The tangential component of the vector field at the mesh edges is used as a degree of freedom. Also called *Nedelec's edge element* or just *edge element*.

Index