AC/DC Module

User's Guide

COMSOL

AC/DC Module User's Guide

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located on the COMSOL Access page at www.comsol.com/support/case. Useful links:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/product-update
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase
- Learning Center: https://www.comsol.com/support/learning-center

Part number: CM020101

Contents

Chapter 1: Introduction

Chapter 2: Theory for the AC/DC Module

Chapter 3: Modeling with the AC/DC Module

Chapter 4: Electric Field and Current Interfaces

Chapter 5: Magnetic Field Interfaces

Chapter 6: The Electrical Circuit Interface

Chapter 7: Multiphysics Interfaces and Couplings

Chapter 8: 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
- Overview of the User's Guide

About the AC/DC Module

In this section:

- What Can the AC/DC Module Do?
- AC/DC Module Physics Interface Guide
- AC/DC Module Study Availability
- Common Physics Interface and Feature Settings and Nodes
- Where Do I Access the Documentation and Application Libraries?

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 resistors, inductors, capacitors, and electrical machinery. With this module you can run static, quasistatic, 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
- Magnetostatics
- Low-frequency electromagnetics

Material properties include inhomogeneous and fully anisotropic materials, media with gains or losses, and complex-valued material properties. Infinite element 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.

Selected material data are provided with the module.

🔍	For detailed information about Materials including the AC/DC Material Library and Nonlinear Magnetic Material Library, see the *COMSOL Multiphysics Reference Manual*.

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 AC/DC interfaces are fully multiphysics enabled — couple them to any other physics interface in COMSOL Multiphysics or other modules. The AC/DC Module contains predefined multiphysics interfaces to facilitate easy setup of models with the most commonly occurring couplings. For example, the Induction Heating multiphysics interface combines all features from the Magnetic Fields interface in the time-harmonic formulation with the Heat Transfer interface to model induction and eddy current heating.

The AC/DC Module also provides interfaces for modeling electrical circuits.

	• Building a COMSOL Multiphysics Model in the *COMSOL Multiphysics Reference Manual*
Q	• AC/DC Module Physics Interface Guide
	• AC/DC Module Study Availability
	• Common Physics Interface and Feature Settings and Nodes
	• Where Do I Access the Documentation and Application Libraries?

AC/DC Module Physics Interface Guide

The 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* **A**, and the *magnetic scalar potential* V_m.

Each physics 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 physics 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 Reference Manual*.

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.

> **!** When using an axisymmetric physics 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.

See 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.

> **Q** In the *COMSOL Multiphysics Reference Manual:*
>
> • Studies and Solvers
> • The Physics Interfaces
> • For a list of all the core physics interfaces included with a COMSOL Multiphysics license, see Physics Interface Guide.

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
⚡ AC/DC				
Electric Currents[1]		ec	all dimensions	stationary; stationary source sweep; frequency domain; time dependent; small signal analysis, frequency domain; eigenfrequency
Electric Currents in Shells		ecis	3D	stationary; frequency domain; time dependent; eigenfrequency
Electric Currents in Layered Shells		ecis	3D	stationary; frequency domain; time dependent; eigenfrequency

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Electrical Circuit		cir	Not space dependent	stationary; frequency domain; time dependent; frequency domain; eigenfrequency
Electrostatics[1]		es	all dimensions	stationary; time dependent; stationary source sweep; eigenfrequency; frequency domain; small signal analysis, frequency domain
Electrostatics, Boundary Elements		esbe	3D, 2D	stationary; stationary source sweep; frequency domain; small signal analysis, frequency domain
Magnetic Fields[1]		mf	3D, 2D, 2D axisymmetric	stationary; frequency domain; time dependent; small signal analysis, frequency domain; coil geometry analysis (3D only); time-to-frequency losses; eigenfrequency
Magnetic Field Formulation		mfh	3D, 2D, 2D axisymmetric	stationary; frequency domain; time dependent; small signal analysis, frequency domain; time-to-frequency losses
Magnetic Fields, No Currents		mfnc	3D, 2D, 2D axisymmetric	stationary; frequency domain; time dependent; time-to-frequency losses
Magnetic Fields, No Currents, Boundary Elements		mfncbe	3D, 2D	stationary
Magnetic Fields, Currents Only		mfco	3D	stationary; stationary source sweep with initialization; frequency domain source sweep with initialization

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Magnetic and Electric Fields		mef	3D, 2D, 2D axisymmetric	stationary; frequency domain; time dependent; stationary source sweep; frequency domain source sweep; coil geometry analysis (3D only)
Rotating Machinery, Magnetic		rmm	3D, 2D	stationary; time dependent, coil geometry analysis (3D only); frequency domain; time-to-frequency losses
Magnetic Machinery, Rotating, Time Periodic		mmtp	2D	stationary; time dependent; frequency domain
Magnetohydrodynamics		—	3D, 2D, 2D axisymmetric	stationary; time dependent; frequency-stationary; frequency-transient
Particle Tracing				
Particle Field Interaction, Nonrelativistic[4]		—	3D, 2D, 2D axisymmetric	bidirectionally coupled particle tracing; time dependent
Particle Field Interaction, Relativistic,[4]		—	3D, 2D, 2D axisymmetric	bidirectionally coupled particle tracing; time dependent
Charged Particle Tracing[4]		cpt	3D, 2D, 2D axisymmetric	bidirectionally coupled particle tracing; time dependent
Heat Transfer				
Electromagnetic Heating				

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Induction Heating[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent; frequency-stationary; frequency-transient; frequency-stationary, one-way electromagnetic heating; frequency-transient, one-way electromagnetic heating; small-signal analysis, frequency domain
Joule Heating[1,2]		—	all dimensions	stationary; time dependent; frequency-transient; small-signal analysis; frequency domain; frequency-stationary; frequency-stationary, one-way electromagnetic heating; frequency-transient, one-way electromagnetic heating
Structural Mechanics				
Thermal–Structure Interaction				
Joule Heating and Thermal Expansion[2,3]		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Electromechanics				
Electromechanics[2,3]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency, prestressed; time dependent; frequency domain, prestressed
Electromechanics, Shell[2,5]			3D, 2D axisymmetric	stationary; eigenfrequency, prestressed; time dependent; frequency domain, prestressed

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Electromechanics, Membrane[2,5]			3D, 2D axisymmetric	stationary; eigenfrequency, prestressed; time dependent; frequency domain, prestressed
Electromechanics, Boundary Elements[2,3]		—	3D, 2D	stationary; eigenfrequency, prestressed; time dependent; frequency domain, prestressed
Piezoelectricity				
Piezoelectricity, Solid[2]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, modal; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; small-signal analysis, frequency domain; linear buckling
Piezoelectricity, Layered Shell[2,7]		—	3D	stationary; eigenfrequency; time dependent; frequency domain
Magnetomechanics				
Piezomagnetism[2,3]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Nonlinear Magnetostriction[2,3]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed
Magnetomechanics[2,3]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed
Magnetomechanics, No Currents[2,3]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed
Magnetic–Elastic Interaction in Rotating Machinery[2,5]		—	3D, 2D	stationary; time dependent; frequency domain;
Magnetic–Rigid Body Interaction in Rotating Machinery[,2,6]		—	3D, 2D	stationary; time dependent; frequency domain;
⤓ **Electrostriction**				
Ferroelectroelasticity[2,3]		—	3D, 2D, 2D axisymmetric	stationary; time dependent; frequency domain
Electrostriction[2,3]		—	3D, 2D, 2D axisymmetric	stationary; time dependent; frequency domain

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
🔧 **Piezoresistivity**				

[1] This physics interface is included with the core COMSOL package but has added functionality for this module.

[2] This physics interface is a predefined multiphysics coupling that automatically adds all the physics interfaces and coupling features required.

[3] Requires the addition of the Structural Mechanics Module or the MEMS Module.

[4] Requires the addition of the Particle Tracing Module.

[5] Requires the addition of the Structural Mechanics Module.

[6] Requires the addition of the Multibody Dynamics Module.

[7] Requires the addition of the Composite Materials 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 [1]				PRESET STUDIES [2]									
			MAGNETIC FIELD	ELECTRIC FIELD	MAGNETIC POTENTIAL	CURRENT DENSITY	STATIONARY	TIME DEPENDENT	FREQUENCY DOMAIN	EIGENFREQUENCY	SMALL-SIGNAL ANALYSIS, FREQUENCY DOMAIN	TIME TO FREQUENCY LOSSES	STATIONARY SOURCE SWEEP W. INITIALIZATION	FREQUENCY-STATIONARY	FREQUENCY-TRANSIENT	FREQ. DOMAIN SOURCE SWEEP W. INITIALIZATION
Electrostatics	es	V		x y z			√	√	√	√	√		√			
Electrostatics, Boundary Elements	esbe	V		x y z			√						√			
Electric Currents	ec	V		x y z		x y z	√	√	√	√	√					
Electric Currents in Shells	ecis	V		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[1]				PRESET STUDIES[2]									
			MAGNETIC FIELD	ELECTRIC FIELD	MAGNETIC POTENTIAL	CURRENT DENSITY	STATIONARY	TIME DEPENDENT	FREQUENCY DOMAIN	EIGENFREQUENCY	SMALL-SIGNAL ANALYSIS, FREQUENCY DOMAIN	TIME TO FREQUENCY LOSSES	STATIONARY SOURCE SWEEP W. INITIALIZATION	FREQUENCY-STATIONARY	FREQUENCY-TRANSIENT	FREQ. DOMAIN SOURCE SWEEP W. INITIALIZATION
Electric Currents in Layered Shells	ecis	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 Field Formulation	mfh	$\mathbf{H}$	x y z				√	√	√		√	√				
Magnetic and Electric Fields	mef	$V, \mathbf{A}$	x y z	x y z	x y z	x y z	√	√	√		√			√	√	
Magnetic Fields, No Currents	mfnc	V_m	x y z				√	√				√				
Magnetic Fields, No Currents, Boundary Elements	mfncbe	V_m	x y z				√									
Magnetic Fields, Currents Only	mfco	$\mathbf{A}$	x y z		x y z	x y z	√							√	√	√
Rotating Machinery, Magnetic	rmm	$\mathbf{A}, V_m$	x y	z	z	z	√	√	√			√				
Magnetic Machinery, Rotating, Time Periodic	mmtp	$\mathbf{A}$	x y	z	z	z	√	√	√							

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

PHYSICS INTERFACE	TAG	DEPENDENT VARIABLES	FIELD COMPONENTS [1]				PRESET STUDIES [2]									
			MAGNETIC FIELD	ELECTRIC FIELD	MAGNETIC POTENTIAL	CURRENT DENSITY	STATIONARY	TIME DEPENDENT	FREQUENCY DOMAIN	EIGENFREQUENCY	SMALL-SIGNAL ANALYSIS, FREQUENCY DOMAIN	TIME TO FREQUENCY LOSSES	STATIONARY SOURCE SWEEP W. INITIALIZATION	FREQUENCY-STATIONARY	FREQUENCY-TRANSIENT	FREQ. DOMAIN SOURCE SWEEP W. INITIALIZATION
Electrical Circuit	cir	not applicable					√	√	√	√						
Induction Heating[3]	—	$\mathbf{A}$,T	x y z	x y z	x y z	x y z	√	√	√					√	√	

[1] 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.

[2] Custom studies (for example, Eigenvalue) are also available based on the physics interface.

[3] It is a multiphysics interface that couples the Magnetic Fields interface and the Heat Transfer interface. Other multiphysics interfaces related to the AC/DC Module are not shown in this table.

Ⓠ
- Studies and Solvers in the *COMSOL Multiphysics Reference Manual*
- What Can the AC/DC Module Do?
- AC/DC Module Physics Interface Guide
- Where Do I Access the Documentation and Application Libraries?

Common Physics Interface and Feature Settings and Nodes

There are several common settings and sections available for the physics interfaces and feature nodes. Some of these sections also have similar settings or are implemented in the same way no matter the physics interface or feature being used.

In each module's documentation, only unique or extra information is included; standard information and procedures are centralized in the *COMSOL Multiphysics Reference Manual*.

> In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Where Do I Access the Documentation and Application Libraries?

A number of online resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or context-based) help, and the Application Libraries are all accessed through the COMSOL Desktop.

> If you are reading the documentation as a PDF file on your computer, the blue links do not work to open an application or content referenced in a different guide. However, if you are using the Help system in COMSOL Multiphysics, these links work to open other modules, application examples, and documentation sets.

THE DOCUMENTATION AND ONLINE HELP

The *COMSOL Multiphysics Reference Manual* describes the core physics interfaces and functionality included with the COMSOL Multiphysics license. This book also has instructions on how to use COMSOL Multiphysics and how to access the electronic Documentation and Help content.

Opening Topic-Based Help

The Help window is useful as it is connected to the features in the COMSOL Desktop. To learn more about a node in the Model Builder, or a window on the Desktop, click to highlight a node or window, then press F1 to open the Help window, which then

displays information about that feature (or click a node in the Model Builder followed by the **Help** button (?). This is called *topic-based* (or *context) help*.

Win	To open the **Help** window: • In the **Model Builder**, **Application Builder**, or **Physics Builder**, click a node or window and then press F1. • On any toolbar (for example, **Home**, **Definitions**, or **Geometry**), hover the mouse over a button (for example, **Add Physics** or **Build All**) and then press F1. • From the **File** menu, click **Help** (?). • In the upper-right corner of the COMSOL Desktop, click the **Help** (?) button.

Mac **Linux**	To open the **Help** window: • In the **Model Builder** or **Physics Builder**, click a node or window and then press F1. • In the main toolbar, click the **Help** (?) button. • From the main menu, select **Help>Help**.

Opening the Documentation Window

Mac **Linux**	To open the **Documentation** window: • Press Ctrl+F1. • In the main toolbar, click the **Documentation** (▮) button. • From the main menu, select **Help>Documentation**.

THE APPLICATION LIBRARIES WINDOW

Each model or application includes documentation with the theoretical background and step-by-step instructions to create a model or application. The models and applications are available in COMSOL Multiphysics as MPH-files that you can open for further investigation. You can use the step-by-step instructions and the actual models as templates for your own modeling. In most models, SI units are used to describe the relevant properties, parameters, and dimensions, but other unit systems are available.

Once the Application Libraries window is opened, you can search by name or browse under a module folder name. Click to view a summary of the model or application and its properties, including options to open it or its associated PDF document.

Opening the Application Libraries Window
To open the **Application Libraries** window ( ):

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

COMSOL ACCESS AND TECHNICAL SUPPORT
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 a case number will be sent to you by email. You can also access technical support, software updates, license information, and other resources by registering for a COMSOL Access account.

COMSOL ONLINE RESOURCES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/product-update
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/forum
Events	www.comsol.com/events
COMSOL Application Gallery	www.comsol.com/models
COMSOL Video Gallery	www.comsol.com/videos
Learning Center	www.comsol.com/support/learning-center
Support Knowledge Base	www.comsol.com/support/knowledgebase

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 Reference Manual*.

> 💡 As detailed in the section Where Do I Access the Documentation and Application Libraries? this information can also be searched 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 FOR THE AC/DC MODULE

The Theory for the AC/DC Module chapter contains an overview of the theory behind the AC/DC Module. It is intended for readers that want to understand what goes on in the background when using the physics interfaces and — in addition to giving the theory background for each of the physics interfaces — discusses the Fundamentals of Electromagnetics, Electromagnetic Forces, and Electromagnetic Quantities.

MODELING WITH THE AC/DC MODULE

The Modeling with the AC/DC Module chapter summarizes general procedures and strategies for modeling low-frequencies electromagnetics. Topics include Preparing for Modeling, Modeling Coils, Computing Forces and Torques, Computing Losses, Computing Lumped Parameters, Connecting to Electrical Circuits.

ELECTRIC FIELD AND CURRENT INTERFACES

The Electric Field and Current Interfaces chapter describes the electric field and current interfaces supported by the AC/DC Module:

- The Electrostatics Interface, which simulates electric fields in dielectric materials with a fixed or slowly-varying charge distribution. Preset Stationary, Stationary Source Sweep, Time Dependent, Frequency Domain, Small-Signal Analysis, and Stationary Source Sweep study types are available.

- The Electrostatics, Boundary Elements Interface, which simulates electric fields in dielectric materials with a fixed charge distribution using the boundary element method. Preset Stationary and Stationary Source Sweep 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. Preset Stationary, Stationary Source Sweep, Time Dependent, Frequency Domain, Small-Signal Analysis, and Stationary Source Sweep study types are available.

- The Electric Currents in Shells Interface, computes electric fields, currents, and potential distributions in thin conducting layered shells. It is used under conditions where inductive effects are negligible; that is, when the skin depth is much larger than the size of studied device. Preset Stationary, Time Dependent, and Frequency Domain study types are available.

- The Electric Currents in Layered Shells Interface, which is very similar to The Electric Currents in Shells Interface.

MAGNETIC FIELD INTERFACES

The Magnetic Field Interfaces chapter describes the magnetic field interfaces supported by the AC/DC Module:

- 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. Thus it is the primary choice for modeling permanent magnets. Preset Stationary and Time Dependent study types are available.

- The Magnetic Fields, No Currents, Boundary Elements Interface, which handles magnetic fields without currents using the boundary element method. It can be combined with the Magnetic Fields or Magnetic Fields, No Currents interfaces for easy modeling of open boundary conditions. The preset Stationary study type is available.

- The Magnetic Fields Interface, which computes magnetic fields with source currents by solving the magnetic vector potential. Preset Stationary, Time Dependent, Frequency Domain, Small-Signal Analysis, and Time to Frequency Losses study types are available. This is the recommended primary choice for modeling magnetic fields involving source currents.

- The Magnetic and Electric Fields Interface computes magnetic field and current distributions when the exciting current is driven by an applied voltage. In most cases, using the Magnetic Fields interface with its dedicated modeling features is the preferred choice over using the Magnetic and Electric Fields interface. Preset Stationary, Time Dependent, Frequency Domain study types are available.

- The Magnetic Field Formulation Interface has the equations, boundary conditions, and currents for modeling magnetic fields, solving for the magnetic field components. It is especially suitable for modeling involving nonlinear conductivity effects, for example in superconductors. Preset Stationary, Time Dependent, Frequency Domain, and Small-Signal Analysis study types are available.

- The Magnetic Fields, Currents Only Interface, which handles magnetic fields with currents only. It is especially suitable for computing the lumped matrix of an array of conductors in free space. Preset Stationary and Stationary Source Sweep with Initialization study types are available.

- The Rotating Machinery, Magnetic Interface combines a Magnetic Fields formulation (magnetic vector potential) and Magnetic Fields, No Currents formulation (magnetic scalar potential) with a selection of predefined frames for prescribed rotation or rotational velocity. Most of its features are taken either from the Magnetic Fields or the Magnetic Fields, No Currents interfaces. Preset Stationary and Time Dependent study types are available.

- The Magnetic Machinery, Rotating, Time Periodic Interface uses a magnetic vector potential formulation, but with only the out-of-plane component. By using a time periodic approach, it is possible to simultaneously solve for all of the time frames. Some of its features are taken from the Magnetic Fields interface. Preset Stationary, Frequency Domain, and Time Dependent study types are available.

THE ELECTRICAL CIRCUIT INTERFACE

The Electrical Circuit Interface chapter describes the physics 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 physics interface is also included. Preset Stationary, Time Dependent and Frequency Domain study types are available.

MULTIPHYSICS INTERFACES AND COUPLINGS

The Multiphysics Interfaces and Couplings chapter includes a detailed description for the predefined multiphysics interfaces supported by the AC/DC Module, for example, the Induction Heating interface. It also presents an overview of other predefined multiphysics interfaces and the multiphysics coupling features related to the AC/DC Module.

2

Theory for the AC/DC Module

This chapter contains an overview of the theory behind the AC/DC Module. It is intended for readers that want 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
- Constitutive Relations
- Boundary Conditions
- Potentials
- Reduced Potential PDE Formulations
- Electromagnetic Energy
- The Quasistatic Approximation and the Lorentz Term
- Material Properties
- Frequency Domain
- Time Periodic
- Nonlinear Permanent Magnet Constitutive Relations
- Effective Nonlinear Magnetic Constitutive Relations
- The Jiles–Atherton Hysteresis Model
- References for Electromagnetic Theory

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 field **D**
- Magnetic field intensity **H**
- Magnetic flux density **B**
- Current density **J**
- Electric charge density ρ

For general time-varying fields, Maxwell's equations in the differential form can be written as

$$\nabla \cdot \mathbf{D} = \rho \qquad \text{Gauss' law, electric}$$

$$\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t} \qquad \text{Faraday's law}$$

$$\nabla \cdot \mathbf{B} = 0 \qquad \text{Gauss' law, magnetic} \tag{2-1}$$

$$\nabla \times \mathbf{H} = \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} \qquad \text{Maxwell-Ampère's law}$$

Another fundamental equation, derived from Maxwell's equations, is the *equation of continuity*:

$$\nabla \cdot \mathbf{J} = -\frac{\partial \rho}{\partial t} \tag{2-2}$$

Constitutive Relations

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

$$\mathbf{D} = \varepsilon_0 \mathbf{E} + \mathbf{P}$$

$$\mathbf{B} = \mu_0 (\mathbf{H} + \mathbf{M}) \tag{2-3}$$

$$\mathbf{J} = \sigma \mathbf{E}$$

where ε_0 and μ_0 are the permittivity and permeability of vacuum, and σ is the electric conductivity. The constants ε_0 and μ_0 are available in COMSOL Multiphysics as predefined physical constants. In the SI unit system, ε_0 has an approximate value of $1/(36\pi) \cdot 10^{-9}$ F/m; μ_0 has an approximate value of $4\pi \cdot 10^{-7}$ H/m.

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 $\mathbf{P}$ 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 $\mathbf{M}$ 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, respectively.

GENERALIZED CONSTITUTIVE RELATIONS

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 $\mathbf{J} = \sigma\mathbf{E} + \mathbf{J}_e$.

ENERGY- AND COENERGY-BASED MAGNETIC CONSTITUTIVE RELATIONS

Another general approach to define the constitutive relation of a magnetic material stems from the conservation of magnetic energy density and magnetic coenergy density. For nonlinear magnetic materials without hysteresis, these densities are defined as follows.

$$dW_m = \int_0^{\mathbf{B}} \mathbf{H} \cdot d\mathbf{B'}$$
$$dW'_m = \int_0^{\mathbf{H}} \mathbf{B} \cdot d\mathbf{H'}$$

In this context, the magnetic field $\mathbf{H}$ can be derived from the magnetic flux density $\mathbf{B}$ according to

$$\mathbf{H} = \frac{\partial dW_m}{\partial \mathbf{B}}$$

Equivalently, the magnetic flux density can be derived from the magnetic field as

$$\mathbf{B} = \frac{\partial dW_m'}{\partial \mathbf{H}}$$

Boundary 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

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

(2-4)

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 two.
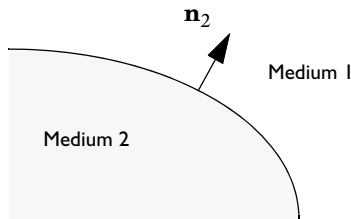
The boundary condition for the current density, derived from Equation 2-4, is expressed as

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

INTERFACE BETWEEN A DIELECTRIC AND A PERFECT CONDUCTOR
A perfect conductor has infinite electric 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 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:

$$
\begin{aligned}
-\mathbf{n}_2 \cdot \mathbf{D}_2 &= \rho_s \\
-\mathbf{n}_2 \times \mathbf{E}_2 &= 0 \\
-\mathbf{n}_2 \cdot \mathbf{B}_2 &= 0 \\
-\mathbf{n}_2 \times \mathbf{H}_2 &= \mathbf{J}_s
\end{aligned}
\tag{2-5}
$$

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 **A**. They are given by the equalities

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

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 is useful for models involving a uniform or known external background field, usually originating from distant sources that might be expensive or inconvenient to include in the model geometry. A typical example is when analyzing induced magnetization in ferromagnetic objects such as ships or vehicles due to Earth's magnetic field. The strategy is then to solve only for the induced fields represented by the reduced vector potential $\mathbf{A}_{red}$, introducing the substitution $\mathbf{A} = \mathbf{A}_{red} + \mathbf{A}_{ext}$, where $\mathbf{A}_{ext}$ represents the known background field, 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 quasistatic systems solving for an *A* formulation, the reduced potential formulation results in the following PDE:

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

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

$$P_e = \int_V \mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} dV$$

$$P_m = \int_V \mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} dV$$

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,

$$P_h = \int_V \mathbf{J} \cdot \mathbf{E} dV$$

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

$$P_r = \oint_S (\mathbf{E} \times \mathbf{H}) \cdot \mathbf{n} dS$$

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

Under the assumption that 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:

$$w = w_e + w_m = \frac{1}{2}\varepsilon\mathbf{E} \cdot \mathbf{E} + \frac{1}{2\mu}\mathbf{B} \cdot \mathbf{B}$$

The Quasistatic 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 *quasistatic 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 quasistatic 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

$$\mathbf{J} = \sigma(\mathbf{E} + \mathbf{v} \times \mathbf{B}) + \mathbf{J}_e$$

where $\mathbf{J}_e$ is an externally generated current density.

Maxwell–Ampère's law for quasistatic systems is consequently extended to

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

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 for which they require some consideration. A given 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 have 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 relations.

MATERIAL PROPERTIES AND THE MATERIAL BROWSER

All interfaces in the AC/DC Module support the use of the COMSOL Multiphysics material libraries. The typical electromagnetic material properties that can be stored are:

• The electric conductivity

- The relative permittivity
- The relative permeability

The physics-specific domain material properties are by default taken from the material specification. The material properties are inputs to material laws or constitutive relations that are defined on the feature level below the physics interface node in the model tree. There is one editable default domain feature that initially represents a linear isotropic material. Domains with different material laws are specified by adding additional features. Some of the domain parameters can either be a scalar or a matrix (tensor) depending on whether the material is isotropic or anisotropic.

In a similar way, boundary, edge, and point settings are specified by adding the corresponding features. A certain feature might require one or several fields to be specified, while others generate the conditions without user-specified fields.

For detailed information about Materials and Modeling Anisotropic Materials, including the Electromagnetic Models, see the *COMSOL Multiphysics Reference Manual.*

Frequency Domain

PHASORS

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

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

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

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

The field $\tilde{\mathbf{E}}(\mathbf{r})$ is a *phasor*, which contains amplitude and phase information of the field but is independent of t.

FREQUENCY DOMAIN FORMULATION

With the use of phasors, Maxwell's equations can be formulated in the frequency domain. One of the main advantages is that a time derivative corresponds to a multiplication by $j\omega$:

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

If the fields satisfy a linear time-dependent equation, then the corresponding phasors must satisfy a similar equation in which the time derivatives are replaced by a factor $j\omega$. In this way, linear differential equations are converted to algebraic equations that are much easier to solve.

For sake of simplicity, when writing variables and equations in the frequency domain, the tilde is dropped from the variable denoting the phasor. However, it is important to remember that the field that has been calculated is a phasor and not a physical field. Also note that the phasor $\tilde{\mathbf{E}}(\mathbf{r})$ is visualized in the plot as $\text{Re}(\tilde{\mathbf{E}}(\mathbf{r}))$ by default, which is $\mathbf{E}$ at time $t = 0$. To obtain the solution at a given time, specify a phase factor in all results settings and in the corresponding functions.

The frequency domain formulation is only applicable for equations linear in the fields and for one specific frequency. In general, it cannot be used with materials whose properties depend on the fields themselves. To model such nonlinear materials with the frequency domain formulation, different approaches are required. See Effective Nonlinear Magnetic Constitutive Relations for a formulation that approximates nonlinear magnetic constitutive relations in time-harmonic problems.

SIGN CONVENTION

The time dependency of a time-harmonic field can be written in two ways: $e^{j\omega t}$ or $e^{-j\omega t}$. In COMSOL Multiphysics, the former sign convention $e^{j\omega t}$ is used.

The time-harmonic sign convention dictates the sign convention for some material coefficients in the frequency domain, such as complex permittivity, complex permeability, and complex refractive index. For example, in the dielectric loss model, the complex relative permittivity is defined as

$$\varepsilon_r = \varepsilon' - j\varepsilon''$$

where ε' and ε'' are two material parameters that are named as the *real part* and *imaginary part* of the relative permittivity, respectively. Inserting the complex relative permittivity in the frequency domain Maxwell–Ampère's law gives

$$\nabla \times \mathbf{H} = j\omega\varepsilon_0\varepsilon'\mathbf{E} + (\omega\varepsilon_0\varepsilon'' + \sigma)\mathbf{E}$$

where the term $\omega\varepsilon_0\varepsilon''$ contributes to energy loss that is indistinguishable from the loss quantified by the electric conductivity σ. With the sign convention used here, a positive material parameter ε'' corresponds to a loss although the imaginary part of the complex

number $\varepsilon' - j\varepsilon''$, evaluated as `imag(ε' – jε'')`, is negative. See Modeling Losses in the Frequency Domain for details.

Time Periodic

Whenever a problem is time periodic, the fields are functions of the phase ϕ as well as periodic with the period 2π. The difference compared to the Frequency Domain is that the dependence is not necessarily harmonic here. A generic periodic phase function is allowed and affects

- Dependent variables, such as the out-of-plane component of the magnetic vector potential $A_Z(\phi)$ in The Magnetic Machinery, Rotating, Time Periodic Interface in 2D.

- Derived quantities, such as the magnetic flux density $\mathbf{B}(\phi) = \nabla \times (A_Z(\phi)\mathbf{e}_Z)$ in The Magnetic Machinery, Rotating, Time Periodic Interface in 2D.

- External sources defined through user inputs, such as the out-of-plane component of the external current density $J_{eZ}(\phi)$ in The Magnetic Machinery, Rotating, Time Periodic Interface in 2D.

The frequency of the time periodic signal is typically referred to as f_{TP} and is a physics variable, which may be inherited from the study. The phase is related to this frequency by the usual relation $\phi = 2\pi f_{TP}t$. It follows that time derivatives in time periodic problems can conveniently be computed as derivatives with respect to phase through the relation

$$\frac{\partial A_Z}{\partial t} = 2\pi f_{TP}\frac{\partial A_Z}{\partial \phi}$$

The phase dependence is typically mapped on evenly distributed points and phase derivatives at a given phase value are computed as a ratio between the variable values at the closest phase values.

Averages, root mean squares, standard deviations, and Fourier transform coefficients derived from time periodic variables are available in result evaluation. For The Magnetic Machinery, Rotating, Time Periodic Interface, more details can be found in Feature Specific Variables and Accessing the Solution at Different Phases.

Nonlinear Permanent Magnet Constitutive Relations

The BH nonlinear permanent magnet constitutive relation available in the **Magnetic Fields** and **Magnetic Fields, No Currents** interfaces is intended for easy modeling of self-demagnetization of soft permanent magnets (for example, AlNiCo). It uses a nonlinear BH-relation that is isotropic around a point in $\mathbf{H}$ space that is shifted by the coercive field $\mathbf{H_c}$. The shifted B-H curve that is specified in the associated material node is thus supplied in the first quadrant; that is, it is shifted by $\mathbf{H_c}$ with respect to the uniaxial curve. The **AC/DC** material library in the **Add Material** window contains an example material, **Nonlinear Permanent Magnet** to be used with this constitutive relation. It is a generic and approximate representation of AlNiCo 5.

Effective Nonlinear Magnetic Constitutive Relations

The effective constitutive relations can be used to approximate the behavior of a nonlinear magnetic material, such as a saturable material, in a (linear) Frequency Domain study.

The approach consists in replacing the nonlinear material with an inhomogeneous linear material — a material described by a magnetic permeability constant in time but which can be space dependent. The local magnetic permeability is chosen using an approximation criterion, such as, for example, that the cycle-average energy stored must be equal to the nonlinear case. As a consequence, the permeability may depend on the amplitude of the magnetic field and the nonlinear solver is invoked during the solution.

The formulation still uses the assumption of harmonic time dependency of the fields (which are still described as phasors): the fields still oscillate at the given frequency; higher-order harmonic effects, or other similar effects due to the nonlinearity, are not accounted for.

Using this constitutive relation provides a better approximation of the behavior of a nonlinear material in the frequency domain than linearizing the material properties, still avoiding the computational cost of a full transient analysis.

The application Effective Nonlinear Magnetic Curves Calculator in the AC/DC Module Application Library can be used to compute the effective B-H curve for a material, starting from its B-H curve.

The Jiles–Atherton Hysteresis Model

Magnetic hysteresis originates from the alignment of electron spins into magnetic domains and the reorganization of such domains under the influence of an externally applied magnetizing field. In the Jiles–Atherton model, for each domain there is a reversible rotation and an irreversible rotation, which corresponds to a reversible and irreversible magnetization. In a magnetic material without hysteresis losses, there is only a reversible part of the magnetization, usually modeled with a relative permeability or with a reversible B-H curve. The Jiles–Atherton hysteresis model includes a nonlinear relationship between the change in magnetization $\mathbf{M}$ and the change in the magnetizing field $\mathbf{H}$. The magnetic flux density $\mathbf{B}$ follows from the relation

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

The Jiles–Atherton model is a phenomenological model based on several parameters. The version available in COMSOL Multiphysics is full vectorial and anisotropic (see Ref. 4 and Ref. 5) and is based on the following five parameters:

- Magnetization reversibility: c_r
- Saturation magnetization: M_s
- Domain wall density: a
- Pinning loss: k_p
- Interdomain coupling: α

These quantities are second-order tensors, something that is not reflected by the equation notation here and in the graphical user interface.

The interdomain coupling parameter is used for defining the effective magnetic field $\mathbf{H}_e$, which is the field that is assumed to affect the magnetic moments on a microscopical level:

$$\mathbf{H}_e = \mathbf{H} + \alpha \cdot \mathbf{M}$$

The saturation magnetization and domain wall density determine the anhysteretic part of the material behavior, which in this implementation follows the Langevin function,

$$L(x) = \coth(x) - 1/x$$

through the formula for anhysteretic magnetization $\mathbf{M}_{\mathrm{an}}$:

$$\mathbf{M}_{\mathrm{an}} = \mathrm{M}_s \cdot L(\mathrm{a}^{-1} \cdot |\mathbf{H}_e|) \frac{\mathbf{H}_e}{|\mathbf{H}_e|}$$

where M_s and a are available as diagonal matrices (Ref. 4). The use of the Langevin function should be interpreted as applied component wise.

The magnetization reversibility, c_r, affects the degree of anhysteretic versus hysteretic behavior. If it is set to unity, the model is purely anhysteretic.

The remaining parameter, k_p, controls the pinning and hysteresis of the magnetic moments.

Given the parameters above, the key equation in the Jiles–Atherton model determines the change in the total magnetization $\mathbf{M}$ caused by a change in the effective magnetic field as

$$d\mathbf{M} = \max(\chi \cdot d\mathbf{H}_e, 0) \frac{\chi}{|\chi|} + c_r d\mathbf{M}_{\mathrm{an}}$$

where the auxiliary vector χ is defined as

$$\chi = k_p^{-1} \cdot (\mathbf{M}_{\mathrm{an}} - \mathbf{M})$$

The Jiles–Atherton hysteresis model is mainly suitable for time-dependent studies, but it also supports stationary parametric simulation. The latter is especially useful if the material is starting from a magnetized configuration that is not the solution of a previous transient study. In the user interface, it is possible to select the initial value of magnetization, in which case the initial magnetic field is set accordingly.

It is implemented for the following physics interfaces: Magnetic Fields (based on vector potential); Magnetic Fields, No Currents (based on scalar potential); and Rotating Machinery, Magnetic (mixed vector and scalar potentials).

The internal state variables are discretized directly in the Gauss points, and an auxiliary variable is added to the same domain also for aiding support to simulations with moving domains.

The order of Gauss-point discretization can be selected in the **Discretization** section under the **Magnetic Field** section. For the Gauss-point discretization order, the default is **Automatic**. This default sets the Gauss-point discretization order depending on the order of the parent physics. If the parent physics variable has order n, the Gauss-point order is $2(n-1)$. It is good practice to test that the results do not change significantly when using a Gauss-point discretization of order $2n$ since a good discretization order depends on the kind of magnetic potential, mesh element shape, and space dimension, rather than the pure mesh size. For an example, see:

> *Vector Hysteresis Modeling*: Application Library path **ACDC_Module/Verifications/vector_hysteresis_modeling**

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.

4. A.J. Bergqvist, "A Simple Vector Generalization of the Jiles–Atherton Model of Hysteresis," *IEEE Transactions on Magnetics*, vol. 32, no. 5, pp. 4213–4215, 1996.

5. J.P.A. Bastos and N. Sadowski, *Magnetic Materials and 3D Finite Element Modeling*, CRC Press 2014.

Formulations, Methods, Technologies

For specific electromagnetic problems, it is always convenient to solve subsets or special cases of *Maxwell's equations*, resulting in different electric, magnetic, or electromagnetic formulations. Based on these formulations, COMSOL Multiphysics has several electromagnetics modules (for instance, the AC/DC Module, RF Module, and Wave Optics Module), as well as several different physics interfaces for each module. A formulation is usually based on a vector field, a potential, or a combination of them. The vector fields and potentials are the dependent variables required to solve. Depending on specific cases, different numerical methods and technologies can also be used, resulting in more variants of the physics interfaces. This section aims to provide an overview of the main formulations, methods, and technologies used by AC/DC interfaces.

In this section:

- The V Formulation
- The D-V Formulation
- The Vm Formulation
- The A Formulation
- The A–V Formulation
- The H Formulation
- The A–Vm Formulation
- The Gauge
- The Curl Element
- The Covariant Formulation
- The Boundary Element Method
- The Layered Shell
- The Circuit Model

The V Formulation

In electrostatics where no magnetic field is present, *Maxwell's equations* (Equation 2-1) reduce to two equations: *Gauss' law* and $\nabla \times \mathbf{E} = \mathbf{0}$. By introducing the *electric potential V* and defining $\mathbf{E} = -\nabla V$, the two equations are reduced to a

single equation since $\nabla \times \nabla V = \mathbf{0}$ always holds. Giving the constitutive relationship $\mathbf{D} = \varepsilon_0 \mathbf{E} + \mathbf{P}$ between the electric displacement field $\mathbf{D}$ and the electric field $\mathbf{E}$, Gauss' law is rewritten as

$$-\nabla \cdot (\varepsilon_0 \nabla V - \mathbf{P}) \;=\; \rho \qquad\qquad (2\text{-}6)$$

where ε_0 is the permittivity of vacuum, $\mathbf{P}$ is the electric polarization vector, and ρ is the space charge density. Equation 2-6 is Poisson's equation, which is easy to solve from the numerical point of view. Compared to using direct representation of the field components or a vector potential, the scalar potential reduces the number of unknowns by a factor 3 and the computational load (for **Direct** solvers) up to an order of magnitude. It is also attractive from a theoretical point of view, as it is generally much easier to find analytical solutions to problems formulated using a scalar potential.

For general cases of static electric field and currents, Equation 2-6 has to be solved together with the *equation of continuity* (Equation 2-2). The V formulation is used by all electric field and current interfaces in the AC/DC Module. Depending on specific situations, different physics interfaces together with different study types are available. Deciding what specific physics interface and study type to use requires a basic understanding of the Charge Relaxation Theory. See Theory for the Electrostatics Interface and Theory for the Electric Currents Interface for more details.

The D-V Formulation

As a counterpoint to solving for the scalar potential, as is done in the The V Formulation, electrostatic problems can be reformulated to solve for the flux. The **D-V Formulation** is a mixed formulation technique that solves for the normal components of the electric displacement field, $\mathbf{D}$. It does this by solving a system of two equations. One for the electric displacement field and a second equation for the electric potential, V. The second equation acts as a constraint on the first equation.

Computing the flux directly produces a high accuracy for the electric displacement field and is particularly advantageous for calculating electromechanical forces and geometries that contain infinitely sharp corners. This does come at the cost of an increased computation load. See Theory for the Electrostatics Interface with the D-V Formulation for more details. The **D-V Formulation** aims to improve the accuracy and convergence of the electric field, while the electric potential, serving as a Lagrange multiplier, has a lower order of accuracy. Thus, it is recommended to focus on using or visualizing the electric field rather than the potential.

The Vm Formulation

In magnetostatics where no electric currents are present, *Maxwell's equations* (Equation 2-1) reduce to two equations: *magnetic Gauss' law* and $\nabla \times \mathbf{H} = \mathbf{0}$. By introducing the *magnetic scalar potential* V_m and defining $\mathbf{H} = -\nabla V_m$, the two equations are reduced to a single equation since $\nabla \times \nabla V_m = \mathbf{0}$ always holds. As it can be seen, this is analogous to The V Formulation. Using the constitutive relation $\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$ between the magnetic flux density $\mathbf{B}$ and the magnetic field $\mathbf{H}$, the magnetic Gauss' law becomes

$$-\nabla \cdot (\mu_0 \nabla V_m - \mu_0 \mathbf{M}) = 0 \tag{2-7}$$

where μ_0 is the permeability of vacuum, $\mathbf{M}$ is the magnetization vector. This formulation, referred to as the V_m formulation, is used in The Magnetic Fields, No Currents Interface and The Magnetic Fields, No Currents, Boundary Elements Interface. The resulting Equation 2-7 is also a Poisson's equation that is easy to solve. Therefore, The Magnetic Fields, No Currents Interface is the primary choice for modeling permanent magnets and domains without currents.

The A Formulation

As can be seen, in The V Formulation and The Vm Formulation, introducing a scalar potential makes it possible to formulate Maxwell's equations into a single Poisson's equation. Maxwell's equations can also be formulated with vector potentials. For example, by introducing the *magnetic vector potential* $\mathbf{A}$ and defining $\mathbf{B} = \nabla \times \mathbf{A}$, the magnetic form of Gauss' law ($\nabla \cdot \mathbf{B} = 0$) is inherently satisfied since $\nabla \cdot \nabla \times \mathbf{A} = 0$ always holds.

The $\mathbf{A}$ formulation is used by The Magnetic Fields Interface and The Magnetic Fields, Currents Only Interface in the AC/DC Module. Depending on specific cases (static, quasistatic, transient, and time harmonic), different study types in different dimensions of The Magnetic Fields Interface are available. For more details, see the Theory for the Magnetic Fields Interface. The $\mathbf{A}$ formulation is the most popular formulation used in low-frequencies electromagnetics. Therefore, The Magnetic Fields Interface is the recommended primary choice for modeling magnetic fields involving source currents. However, other formulations are also possible and are useful for specific circumstances.

The A–V Formulation

The A Formulation can be combined with The V Formulation to take into account static electric fields. This formulation, referred to as the **A**–*V* formulation, is used by The Magnetic and Electric Fields Interface. The **A**–*V* formulation solves two dependent variables — **A** and *V* in a fully coupled way. For more details, see the Theory for the Magnetic and Electric Fields Interface.

The H Formulation

For some specific cases, the vector field formulation can be useful. One example is the modeling of superconducting materials that have highly nonlinear electric conductivity. In this case, it is convenient to use the *magnetic field formulation* since the nonlinear constitutive relation between the electric field **E** and the current density **J** can be included naturally. This formulation, referred to as the **H** formulation, is used by The Magnetic Field Formulation Interface. Different from The A Formulation which solves *Ampère's law*, The H Formulation solves *Faraday's law*. For more details, see the Theory for the Magnetic Field Formulation Interface.

The A–Vm Formulation

Mixed formulations are also used in computational electromagnetics. These formulations use different formulations that are more suitable for specific computation domains. One of the most famous mixed formulations is the **A**–V_m formulation, which is a combination of The A Formulation and The Vm Formulation. In this *mixed formulation*, The A Formulation is used to model current-carrying domains, and The Vm Formulation is used to model the air gap and other nonconducting domains. The two formulations are coupled on their common interfacing boundaries. The **A**–V_m formulation is used by The Rotating Machinery, Magnetic Interface.

The Gauge

The electromagnetic fields have unique solutions as determined by Maxwell's equations. However, when substituting the scalar and vector potentials into Maxwell's equations, those potentials typically have an infinite number of solutions — unless additional equations or conditions are applied. For example, in The V Formulation, *V* is not uniquely defined as *V'* = *V* + *C* (*C* is a constant) yields the same electrostatic field; in The A Formulation, **A** also has infinite solutions since **A'** = **A** + ∇*f* (*f* is a scalar field) yields the same magnetic flux density; in The A–V Formulation, **A** and *V* have

infinite solutions due to Gauge Transformations. Therefore, potentials based formulations of Maxwell's equations are preliminarily ungauged. An ungauged formulation can only be solved with an iterative solver as the assembled matrix is singular.

In order to have a unique solution for potentials and be solved with a direct solver, a gauge has to be applied. In The V Formulation, V needs at least one additional condition fixing its general level. Most often, this is provided as a boundary condition (ground or applied potential). In The A Formulation and The A–V Formulation, additional constraint on the divergence of **A** is required. A common choice is the Coulomb gauge $\nabla \cdot \mathbf{A} = 0$.

The Curl Element

Whenever solving for more than a single vector component, it is generally not possible to use Lagrange elements for electromagnetic modeling. The reason is that they force the fields to be continuous everywhere. This implies that the physics interface conditions, which specify that the normal components of the electric fields and the tangential components of the magnetic fields are discontinuous across interior boundaries between media with different permittivity and permeability, cannot be fulfilled. To overcome this problem, The A Formulation, The A–V Formulation, and The A–Vm Formulation based interfaces use the *curl element*, which does not have this limitation. The curl element is also called vector element or edge element.

The solution obtained when using curl elements also better fulfills the divergence conditions $\nabla \cdot \mathbf{D} = \rho$ and $\nabla \cdot \mathbf{B} = 0$ than when using Lagrange elements.

	For more information about the curl element, read the blog post: www.comsol.com/blogs/what-is-the-curl-element-and-why-is-it-used/

The Covariant Formulation

In 2D axisymmetric formulation using vector field such as the magnetic vector potential, it is beneficial to formulate the out-of-plane dependent variable as $\Psi = rA_{\varphi}$, referred to as the covariant formulation. The covariant formulation has better performance in terms of numerical stability and accuracy. Moreover, the covariant dependent variable usually has a physical meaning, for instance, the difference Ψ between two magnetic surfaces is equal to the poloidal magnetic flux per radian.

One of the advantages of Ψ formulation is that for linear discretization it gives more accurate results than that with The A Formulation. However, using linear discretization is not recommended since the magnetic field is zero at the axis. Always perform a mesh adaptation study to make sure the results are not mesh dependent when linear discretization is used.

The Boundary Element Method

The *boundary element method* is complementary to the finite element method, which most of AC/DC interfaces are solved with. The boundary element method is sometimes more efficient than other methods, including finite elements, in terms of computational resources for problems where there is a small surface-to-volume ratio. Physics interfaces based on the boundary element method differ from those based on the finite element method in that they only use mesh elements on the boundaries of the modeled regions (curves in 2D and surfaces in 3D). For a detailed discussion, see Theory for the Boundary Elements PDE in the *COMSOL Multiphysics Reference Manual*.

In the AC/DC Module, The Electrostatics, Boundary Elements Interface and The Magnetic Fields, No Currents, Boundary Elements Interface use the boundary element method.

The Layered Shell

The *layered shell* is a technology that models a 3D geometry using a surface (2D) geometry and an extra dimension (1D) geometry in the through-thickness (or normal) direction. The layered shell technology is sometimes more efficient in terms of computational resources when modeling thin structures such as PCB coils. In a layered shell, the surface geometry is a physical geometry and supposed to be created in the model whereas the extra dimension geometry is a virtual geometry created by the Layered Material and similar nodes. For more details, see the Layered Material in the *COMSOL Multiphysics Reference Manual*.

In the AC/DC Module, The Electric Currents in Shells Interface and The Electric Currents in Layered Shells Interface as well as some physics features use the layered shell technology.

The Circuit Model

The AC/DC Module provides The Electrical Circuit Interface to solve circuit models. This interface is very useful when a physics field model is coupled with an electrical circuit. For more details, see Theory for the Electrical Circuit Interface.

	Use the **Circuit Extractor** add-in, available in Add-in Library->AC/DC Module, to extract circuit models from physics models.

Theory for the Electrostatics Interface

The Electrostatics Interface is available for 3D, 2D in-plane, and 2D axisymmetric components. Applications with electrostatic 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, that is short compared to the charge relaxation time; also, 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.

In this section:

- Electrostatics Equations
- The Electrostatics Interface in Time Dependent or Frequency Domain Studies
- Effective Material Properties in Effective Media and Mixtures
- Effective Conductivity in Effective Media and Mixtures
- Effective Relative Permittivity in Effective Media and Mixtures
- Effective Relative Permeability in Effective Media and Mixtures
- Ferroelectricity
- Theory for the Electrostatics Interface with the D-V Formulation
- Dispersion Theory
- Archie's Law Theory

	For a discussion about the boundary element method used with The Electrostatics, Boundary Elements Interface, see Theory for the Boundary Elements PDE in the *COMSOL Multiphysics Reference Manual*.

Charge Relaxation Theory

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.

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* for the conduction current density

$$\mathbf{J}_c = \sigma \mathbf{E}$$

the *equation of continuity*

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \mathbf{J}_c = 0$$

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 the 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 2-1 below,

TABLE 2-1: SUITABLE PHYSICS INTERFACE AND STUDY TYPE FOR DIFFERENT TIME-SCALE REGIMES.

CASE	PHYSICS INTERFACE	STUDY TYPE
$\tau \gg t$	Electrostatics	Stationary
$\tau \ll t$	Electric Currents	Stationary
$\tau \sim t$	Electric Currents	Time Dependent or Frequency Domain

FIRST CASE: $\tau \gg T$

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 a given model input. 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 Electric Discharge Module, the Plasma Module, the Semiconductor Module, and the Chemical Reaction Engineering Module.

SECOND CASE: $\tau \ll T$

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 of Ohm's law than above states that

$$\mathbf{J}_c = \sigma \mathbf{E} + \mathbf{J}_e$$

where $\mathbf{J_e}$ is an externally generated current density. The static form of the equation of continuity then reads

$$\nabla \cdot \mathbf{J}_c \ = \ -\nabla \cdot (\sigma \nabla V - \mathbf{J}_e) = \ 0$$

To handle current sources, the equation can be generalized to

$$-\nabla \cdot (\sigma \nabla V - \mathbf{J}_e) = \ Q_j$$

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

$$\nabla \cdot \mathbf{J}_c \ = \ \nabla \cdot (\sigma \mathbf{E} + \mathbf{J}_e) \ = \ -j\omega\rho$$

with the equation ($\nabla \cdot \mathbf{D} = \rho$) yields the following equation for the frequency domain study type:

$$-\nabla \cdot ((\sigma + j\omega\varepsilon_0)\nabla V - (\mathbf{J}_e + j\omega\mathbf{P})) \ = \ 0$$

For the Time Dependent study type, use the transient equation of continuity

$$\nabla \cdot \mathbf{J}_c \ = \ \nabla \cdot (\sigma \mathbf{E} + \mathbf{J}_e) \ = \ -\frac{\partial\rho}{\partial t}$$

and the resulting equation becomes

$$-\nabla \cdot \frac{\partial}{\partial t}(\varepsilon_0 \nabla V + \mathbf{P}) - \nabla \cdot (\sigma \nabla V - \mathbf{J}_e) \ = \ 0$$

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 sometimes does 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.

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 physics 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 physics 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.

The Electrostatics Interface in Time Dependent or Frequency Domain Studies

The Electrostatics Interface can also be solved in a dynamic study (Time Dependent or Frequency Domain). The equation system solved, however, is typically always the one presented in the previous section for the stationary case, in which no transient

electromagnetic effects are taken into account. The difference is that the sources of the problem (charge densities, electric potential) are assumed to be time-varying (in a Time Dependent study) or time-harmonic (in a Frequency Domain study).

> [!] Users that have a license supporting the piezoelectric material model can also include conduction losses in the frequency domain.

The support for dynamic studies simplifies the coupling of the Electrostatics interface with other physics interfaces. Using the physics interface in a dynamic study is a valid approximation only if the time-scale (or the frequency) of the study is so slow that transient electromagnetic effects can be neglected; for example, in acoustic or structural problems.

Most of the features in the Electrostatics interface have the same behavior in all study types, with the important exception of the Terminal node.

The Electrostatics interface also supports the small-signal analysis study sequence, which can be used when a time-harmonic perturbation is superposed on a static bias charge or voltage.

Ferroelectricity

The ferroelectricity phenomenon is related to phase transitions in materials. In its ferroelectric phase, the material exhibits spontaneous polarization, so that it is constituted of domains with nonzero polarization even at zero applied field. This is similar to permanent magnetism in ferromagnetics, which explains the name used for such materials. Application of an electric field can rearrange the domains resulting into the net polarization in the material. At very large electric fields, the polarization saturates, as all ferroelectric domains in the material are aligned along the direction of the applied field. Domain wall interactions can also lead to a significant hysteresis in the polarization.

All domains are assumed to have polarization of the same magnitude P_s, but the polarization can have different orientations. The applied electric field changes the domain orientation, and the resulting net polarization in the ferroelectric material is found from the following nonlinear implicit relation:

$$\mathbf{P} = P_s L(|\mathbf{E}_{\text{eff}}|) \frac{\mathbf{E}_{\text{eff}}}{|\mathbf{E}_{\text{eff}}|} \tag{2-8}$$

The effective field in the material is given by

$$\mathbf{E}_{\text{eff}} = \mathbf{E} + \alpha\mathbf{P}$$

where the matrix α characterizes the interdomain coupling.

The polarization shape is characterized by the function L with the following properties:

For weak effective fields, the polarization is nearly linear

$$\mathbf{P} \approx \chi_0\mathbf{E}_{\text{eff}}$$

and can be characterized by the initial electric susceptibility matrix χ_0.

For strong fields, the polarization magnitude approaches the saturation value

$$\mathbf{P} \approx P_s\frac{\mathbf{E}_{\text{eff}}}{|\mathbf{E}_{\text{eff}}|}$$

Three possible choices are the Langevin function

$$L = \coth\left(\frac{3\chi_0|\mathbf{E}_{\text{eff}}|}{P_s}\right) - \frac{P_s}{3\chi_0|\mathbf{E}_{\text{eff}}|}$$

and a hyperbolic tangent:

$$L = \tanh\left(\frac{\chi_0|\mathbf{H}_{\text{eff}}|}{P_s}\right)$$

HYSTERESIS MODELING

The Jiles–Atherton hysteresis model for ferroelectric materials is available COMSOL Multiphysics. The model assumes that the total polarization can be represented as a sum of reversible and irreversible parts

$$\mathbf{P} = \mathbf{P}_{\text{rev}} + \mathbf{P}_{\text{irr}}$$

The polarization change can be computed from the following incremental equation:

$$d\mathbf{P} = c_r d\mathbf{P}_{\text{an}} + (\mathbf{I} - c_r)d\mathbf{P}_{\text{irr}}$$

where the reversibility is characterized by the matrix c_r, and the anhysteretic polarization is found from a relation similar to Equation 2-8:

$$\mathbf{P}_{an} = P_s L(|\mathbf{E}_{eff}|)\frac{\mathbf{E}_{eff}}{|\mathbf{E}_{eff}|}$$

The change of the irreversible polarization can be computed from

$$d\mathbf{P}_{irr} = \max(\zeta \cdot d\mathbf{E}_{eff}, 0)\frac{\zeta}{|\zeta|}$$

$$\zeta = k_p^{-1}(\mathbf{P}_{an} - \mathbf{P}_{irr})$$

where the pinning loss is characterized by the matrix k_p.

REFERENCE

6. R.C. Smith and C.L. Hom, "A Domain Wall Model for Ferroelectric Hysteresis," *Journal of Intelligent Material Systems and Structures*, vol. 10, no. 3, pp. 195–213, 1999.

Dispersion Theory

Ampere's law is written as

$$\nabla \times \mathbf{H} = \mathbf{J} \qquad (2\text{-}9)$$

The total current is given by

$$\mathbf{J} = \mathbf{J}_i + \frac{\partial \mathbf{D}}{\partial t} + \mathbf{J}_p \qquad (2\text{-}10)$$

where

$$\mathbf{J}_i = \sigma \mathbf{E}$$

is the induction current,

$$\mathbf{D} = \varepsilon_0 \varepsilon_\infty \mathbf{E}$$

is the electric displacement in the high-frequency limit (in the sense that the excitation duration is smaller than the shortest charge relaxation time).

The charge relaxation is modeled using the polarization current (Ref. 1)

$$\mathbf{J}_{\mathrm{p}} = \sum_{m=1}^{N} \mathbf{J}_m$$

DEBYE DISPERSION MODEL

Each term contribution to the total polarization current obeys an equation of the following form:

$$\tau_m \frac{\partial \mathbf{J}_m}{\partial t} + \mathbf{J}_m = \varepsilon_0 \Delta \varepsilon_m \frac{\partial \mathbf{E}}{\partial t} \tag{2-11}$$

where τ_m is the relaxation time, and $\Delta \varepsilon_m$ is the relative permittivity contribution. Note that both quantities can be taken as diagonal matrices to cover the anisotropic case.

Introduce new variables, auxiliary electric field from

$$\mathbf{J}_m = \varepsilon_0 \Delta \varepsilon_m \frac{\partial \mathbf{e}_m}{\partial t}$$

Equation 2-11 can be integrated once with respect to time assuming zero initial fields

$$\tau_m \frac{\partial \mathbf{e}_m}{\partial t} + \mathbf{e}_m = \mathbf{E} \tag{2-12}$$

The polarization current can be expressed the as either

$$\mathbf{J}_{\mathrm{p}} = \frac{\partial}{\partial t} \sum_{m=1}^{N} \varepsilon_0 \Delta \varepsilon_m \mathbf{e}_m \tag{2-13}$$

or

$$\mathbf{J}_{\mathrm{p}} = \varepsilon_0 \sum_{m=1}^{N} \tau_m^{-1} \Delta \varepsilon_m (\mathbf{E} - \mathbf{e}_m)$$

By applying the divergence operator, Equation 2-9 yields the current conservation equation:

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

Assume that the electric conductivity is small, so that the induction current can be neglected. Using Equation 2-13, the current conservation equation can be written as

$$\nabla \cdot \left(\varepsilon_0 \varepsilon_\infty \frac{\partial \mathbf{E}}{\partial t} + \frac{\partial}{\partial t} \sum_{m=1}^{N} \varepsilon_0 \Delta \varepsilon_m \mathbf{e}_m \right) = 0$$

which can be integrated once with respect to time assuming zero initial fields to give the charge conservation equation

$$\nabla \cdot \left(\varepsilon_0 \varepsilon_\infty \mathbf{E} + \sum_{m=1}^{N} \varepsilon_0 \Delta \varepsilon_m \mathbf{e}_m \right) = 0 \qquad (2\text{-}14)$$

Equation 2-14 can be solved using the scalar electric potential V as a dependent variable determine the electric field as $\mathbf{E} = -\nabla V$, and it needs to be solved together with N vector Equation 2-12 for N vector dependent variables $\mathbf{e}_m$.

FREQUENCY DOMAIN

In frequency domain, Equation 2-12 for the auxiliary variables can be solved analytically, so that Equation 2-14 gives

$$\nabla \cdot \left(\varepsilon_0 \varepsilon_\infty \mathbf{E} + \sum_{m=1}^{N} \varepsilon_0 \Delta \varepsilon_m (\mathbf{I} + j\omega\tau_m)^{-1} \mathbf{E} \right) = 0$$

where ω is the angular frequency. By separating the real and imaginary parts, the equation can be further rewritten as

$$\nabla \cdot (\varepsilon_0 \varepsilon_r(\omega)\mathbf{E}) = 0$$

where

$$\varepsilon_r = \varepsilon' - j\varepsilon''$$

with

$$\varepsilon' = \varepsilon_\infty + \sum_{m=1}^{N} \Delta \varepsilon_m (\mathbf{I} + (\omega\tau_m)^2)^{-1}$$

and

$$\varepsilon'' = \sum_{m=1}^{N} \Delta\varepsilon_m \omega \tau_m (I + (\omega\tau_m)^2)^{-1}$$

Each term in the sums in the above equations represents a pole on the complex w plane. Thus, the model is referred to as Multipole Debye Dispersion model. To determine the number of poles needed to approximate the lossy material behavior can be a difficult task. One possible approach is presented in Constant Loss Tangent section.

LOCAL TIME INTEGRATION

In time domain, the auxiliary electric field variables can be treated as the local state variables, and each of N corresponding ODE equations can be integrated locally. Such algorithm makes the overall solution more efficient and also leaner in terms of memory usage. The implemented algorithm is schematically described in the following.

Consider the time evolution of one pole in the Multipole Debye model characterized by an ODE of the type

$$\tau_m \dot{\mathbf{e}}_m + \mathbf{e}_m = \mathbf{E}(t) \tag{2-15}$$

The exact solution to Equation 2-15 is then

$$\mathbf{e}_m^{n+1} = \exp(-\Delta t / \tau_m)\mathbf{e}_m^n + \frac{1}{\tau_m} \int_{t^n}^{t^{n+1}} \exp\left(-\frac{t^{n+1}-s}{\tau_m}\right)\mathbf{E}(s)ds \tag{2-16}$$

By assuming that the electric field $\mathbf{E}(t)$ varies linearly within time increments, the integral in Equation 2-16 can be computed analytically, so the auxiliary electric field at increment $n + 1$ reads

$$\mathbf{e}_m^{n+1} = \mathbf{E}^{n+1} - \exp\left(-\frac{\Delta t}{\tau_m}\right)(\mathbf{E}^n - \mathbf{e}_m^n) - \frac{1 - \exp(-\Delta t / \tau_m)}{-\Delta t / \tau_m}(\mathbf{E}^{n+1} - \mathbf{E}^n) \tag{2-17}$$

All variables in Equation 2-17 are evaluated at the increment n and are stored as internal state variables. Equation 2-17 is solved for each pole of the Multipole Debye model, or for the single pole in the Debye model.

EIGENFREQUENCY STUDY

Direct use of the complex frequency-dependent permittivity would lead to a nonlinear eigenvalue problem, for which the nonlinear eigenvalue solver needs to be used.

Instead, one can solve N algebraic equations for the auxiliary variables $\mathbf{e}_m$

$$j\omega\tau_m\mathbf{e}_m + \mathbf{e}_m = \mathbf{E} \qquad (2\text{-}18)$$

together with Equation 2-14. Even though this approach requires extra degrees of freedom, it will produce frequency independent contributions to the damping and stiffness matrices. The corresponding linear damped eigenvalue problem can be solved using the default solver in a single run for any specified number of eigenfrequencies which will be computed exactly.

> [!] In practice, an Eigenfrequency analysis makes sense only when the Electrostatics interface is used together with some other physics that defines a proper mass matrix. For example, a multiphysics model that involves a Solid Mechanics (or other structural interfaces available in the Structural Mechanics Module). Another possibilities is a coupling between the Electrostatic Interface and electrical circuits.

CONSTANT LOSS TANGENT

In frequency domain, the loss tangent is defined as

$$\eta = \tan\delta = \varepsilon''/\varepsilon' \qquad (2\text{-}19)$$

and it is often used to characterize lossy materials based on experimental data.

In Ref. 2, a special parameterization was suggested for the Multipole Debye model based on the original result derived in Ref. 3 for equivalent electrical circuits. It allows to fit the model parameters for Multipole Debye Model so that the loss tangent is nearly constant at a certain frequency range.

The input parameters are the loss tangent $\eta(f_c)$ together with the center frequency f_c, at which is has been measured, and the model bandwidth n_d that defines a frequency interval (in decades) centered at f_c, in which the loss tangent will be approximately constant and equal to $\eta(f_c)$.

The relaxation times are computed as $\tau_m = 1/\omega_m$, where the corresponding frequencies are equidistantly spaced in the logarithmic space as

$$\omega_m = \omega_c 10^{(N+1-2m)/2}$$

where N is the number of Debye poles, and $\omega_c = 2\pi f_c$.

The corresponding relative permittivity contribution is given by

$$\Delta\varepsilon_m = \varepsilon_\infty(k-1)k^{m-1}$$

where $k > 1$ is the spacing parameter. Note that the relative electric permittivity in the low and high-frequency limits are related simply as $\varepsilon_r(f \to 0) = \varepsilon_\infty k^N$.

COMSOL Multiphysics software will automatically deduce the necessary number of poles N together with the values of the relaxation times τ_m and relative permittivity contributions $\Delta\varepsilon_m$, which will be used in computations to maintain the requested bandwidth and accuracy.

The accuracy can be selected as normal to give

$$\eta(f) = \eta(f_c) \pm 6\%$$

for

$$10^{-n_d/2} < f/f_c < 10^{n_d/2}$$

which requires $N = |n_d| + 3$ poles, and the spacing parameter $k = 10^{2\delta/\pi}$ with the loss angle d computed from $\eta(f_c)$.

Alternatively, the accuracy can be selected as high to get

$$\eta(f) = \eta(f_c) \pm 2\%$$

in the same interval, which will require $N = |3n_d/2| + 5$ poles, and the spacing parameter $k = 10^{4\delta/3\pi}$.

Thus computed parameters will be used automatically in a Time Dependent study using Equation 2-12, and in an Eigenfrequency study using Equation 2-18.

⚠️ In Eigenfrequency study and Time Dependent study, each pole will require one extra vector degree of freedom (domain variable) to represent the corresponding contribution to the polarization. Thus, using many poles in the relaxation data, or choosing high accuracy in case of a constant loss tangent, can lead to significant computation costs for larger models.

FRACTIONAL ORDER DISPERSION MODELS

The Debye dispersion model can be generalized to allow more accurate fitting of the experimental data (Ref. 5).

Cole–Cole Model

The frequency-dependent relative permittivity is modeled in the frequency domain as

$$\varepsilon_r(\omega) = \varepsilon_\infty + \frac{\Delta\varepsilon_r}{1 + (j\omega\tau)^{1-\alpha}}$$

where $\Delta\varepsilon_r$ represents the difference between the low- and high-frequency limiting permittivity values, τ is the relaxation time, and

$$0 \leq \alpha < 1$$

is a complementary broadening parameter.

Havriliak–Negami Model

The frequency-dependent relative permittivity is modeled in the frequency domain as

$$\varepsilon_r(\omega) = \varepsilon_\infty + \frac{\Delta\varepsilon_r}{(1 + (j\omega\tau)^\beta)^\gamma}$$

where $\Delta\varepsilon_r$ represents the difference between the low- and high-frequency limiting permittivity values, τ is the relaxation time, and

$$0 < \beta \leq 1$$

$$0 < \gamma \leq 1/\beta$$

are, respectively, symmetric and asymmetric broadening parameters.

For both Cole–Cole and Havriliak–Negami models, the corresponding polarization current can be computed in the frequency domain as

$$\mathbf{J}_p = \frac{\varepsilon_0\Delta\varepsilon_r}{\tau}(\varepsilon_r(0) - \varepsilon_r(\omega))\mathbf{E}$$

Approximation for Time Domain and Eigenfrequency

The formulation of Cole–Cole and Havriliak–Negami models in time domain requires the use of fractional derivatives. Furthermore, when used in an eigenfrequency analysis, the corresponding fractional order differential equations will always lead to a nonlinear eigenvalue problem.

As an alternative, COMSOL Multiphysics approximates the relative permittivity in the frequency domain as

$$\varepsilon_r(\omega) \approx \varepsilon_\infty + \sum_{m=1}^{N} \frac{\Delta\varepsilon_m}{1+j\omega\tau_m}$$

where the relaxation times τ_m and the corresponding relative permittivity contributions $\Delta\varepsilon_m$ are computed for a given frequency range and accuracy using the partial fraction fitting algorithm. Such approach allows to reuse the implementation of the Multipole Debye model for computations is time domain and for eigenfrequency analysis.

TEMPERATURE EFFECTS

For many dielectric materials such as polymers, the charge relaxation properties have a strong dependence on the temperature. A common assumption is that a change in the temperature can be transformed directly into a change in the time scale. The reduced time is defined as

$$t_r = \int_0^t \frac{dt'}{\alpha_T(T(t'))}$$

where $\alpha_T(T)$ is a temperature-dependent shift function.

The implication is that the problem can be solved using the original material data, provided that the time is transformed into the reduced time. Think of the shift function $\alpha_T(T)$ as a multiplier to the relaxation time in Equation 2-12 and Equation 2-18.

Vogel–Fulcher Shift
The shift function is given by

$$\log(\alpha_T) = \frac{Q}{R(T-T_0)}$$

here, a base-e logarithm is used, Q is the activation energy (SI unit: J/mol), and R is the universal gas constant. Temperature T_0 is the temperature at which the relaxation time would becomes infinite.

Arrhenius Shift
The shift function is assumed to follow an Arrhenius law:

$$\log(\alpha_T) = \frac{Q}{R}\left(\frac{1}{T} - \frac{1}{T_{ref}}\right)$$

here, a base-e logarithm is used, Q is the activation energy (SI unit: J/mol), and R is the universal gas constant.

Williams–Landel–Ferry Shift

One commonly used shift function is defined by the WLF (Williams–Landel–Ferry) equation

$$\log(\alpha_T) = \frac{-C_1(T - T_{\text{ref}})}{C_2 + (T - T_{\text{ref}})}$$

where a base-10 logarithm is used. This shift is only valid over a certain temperature range, typically around the glass transition temperature.

The first step to compute the shift function $\alpha_T(T)$ consists of building a master curve based on experimental data. To do this, the curves of the dielectric properties versus time or frequency are measured at a reference temperature T_{ref}. Then, the same properties are measured at different temperatures.

The shift value of each curve, with respect to the master curve obtained at the temperature T_{ref}, defines the shift factor $\alpha_T(T)$. The constants C_1 and C_2 are material dependent and are calculated after plotting $\log(\alpha_T)$ versus $T - T_{\text{ref}}$.

	The shift factor at the reference temperature equals $\alpha_T(T_{\text{ref}}) = 1$, so that T_{ref} is the temperature at which the master curve is given. If the temperature T drops below $T_{\text{ref}} - C_2$, the WLF equation is no longer valid.

Since the master curve is measured at an arbitrary reference temperature T_{ref}, the shift factor $\alpha_T(T)$ can be derived with respect to any temperature, and it is commonly taken as the shift with respect to the glass transition temperature.

Tool–Narayanaswamy–Moynihan Shift

Charge relaxation in glass can be modeled using the so-called Tool–Narayanaswamy–Moynihan shift factor which is given as

$$\log(\alpha_T) = \frac{Q}{R}\left(\frac{\chi}{T} + \frac{1-\chi}{T_{\text{f}}} - \frac{1}{T_{\text{ref}}}\right)$$

here, a base-e logarithm is used, Q is the activation energy (SI unit: J/mol), R is the universal gas constant, T is the current temperature, T_{ref} is a reference temperature, χ

is a dimensionless activation energy fraction, and T_f is the so-called fictive temperature. The fictive temperature is given as the weighted average of partial fictive temperatures.

$$T_f = \sum_i w_i T_{fi} \text{ with } \sum_i w_i = 1$$

Here, w_i are the weights and T_{fi} are the partial fictive temperatures. The partial fictive temperatures are determined from a system of coupled ordinary differential equations (ODEs) which follow Tool's equation

$$\frac{d}{dt}T_{fi} = \frac{T - T_{fi}}{\alpha_T \lambda_{0i}}$$

here, λ_{0i} is a relaxation time parameter.

REFERENCES FOR DISPERSION

1. N.S. Stoykov, T.A. Kuiken, M.M. Lowery, and A. Taflove, "Finite-element time-domain algorithms for modeling linear Debye and Lorentz dielectric dispersions at low frequencies," *IEEE Trans. Biomed.*, vol. 50, no. 9, pp. 1100–1107, 2003.

2. A.E. Engin, I. Ndip, K. Lang, and J. Aguirre, "Closed-Form Multipole Debye Model for Time-Domain Modeling of Lossy Dielectrics," *IEEE Trans. Electromagn. Compat.*, vol. 61, no. 3, pp. 966–968, 2019.

3. R. Morrison, "RC constant-argument driving-point admittances," *IRE Trans. Circuit Theory*, vol. 6, no. 3, pp. 310–317, 1959.

4. J.A. Diego, J. Sellares, A. Aragoneses, M. Mudarra, J.C. Canadas, and J. Belana, "TSDC study of the glass transition: correlation with calorimetric data," *J. Phys. D: Appl. Phys.*, vol. 40, pp. 138–1145, 2007.

5. A. Schönhals and F. Kremer, "Analysis of Dielectric Spectra," pp. 59–98 in *Broadband Dielectric Spectroscopy*, Springer-Verlag, Berlin, 2003.

Theory for the Electrostatics Interface with the D-V Formulation

The **Electrostatics** Interface has an option to use The D-V Formulation. This is accessed through the discretization options in the electrostatic interface settings. This formulation is available for 2D and 3D components. The difference with this formulation compared to the default **V formulation** used in **Electrostatics** is that instead of solving only for the **Electric potential**, V, the **D-V formulation** is a mixed formulation

method that simultaneously solves for two dependent variables. These are the flux term, the electric displacement field $\mathbf{D}$, as well as the scalar potential, V.

Computing the flux directly is advantageous for applications where the accuracy of the electric displacement field is important. This is useful for situations that involve large fields in tight geometries, such as comb drives in **MEMS** devices. Electromagnetic forces are also derived from the flux term, which makes accurate fluxes important in electromechanical applications.

The standard **Electrostatics** formulation is based on the equations

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

$$\mathbf{E} = -\nabla V$$

where

$$\mathbf{D} = \varepsilon_0 \varepsilon_r \mathbf{E}$$

Having the scalar **Electric potential** V as the dependent variable reduces the system to one equation to be solved. This is computationally straightforward. However, as the electric field, and in turn the electric displacement field, are derived by differentiating this scalar potential, these terms will be expressed as a lower order polynomial and so have less accuracy.

The **D-V formulation** is a mixed formulation that solves two equations and is developed in a way to set $\mathbf{D}$ as a dependent variable along with V. This is achieved using the weak form (see COMSOL documentation on weak form), defining the test functions $\tilde{\mathbf{D}}$ and $\tilde{V}$, and integrating the above equations over a domain Ω to get

$$\int_\Omega (\nabla \cdot \mathbf{D}\tilde{V}) - \int_\Omega \rho\tilde{V} = 0$$

$$\int_\Omega E\tilde{\mathbf{D}} = -\int_\Omega \nabla V\tilde{\mathbf{D}}$$

where the latter can be integrated by parts to yield

$$\int_\Omega V(\nabla \cdot \tilde{\mathbf{D}}) - \int_{\delta\Omega} (\mathbf{n} \cdot \tilde{\mathbf{D}})V - \int_\Omega E\tilde{\mathbf{D}} = 0$$

with $\mathbf{n}$ being the normal vector on the boundary $\delta\Omega$.

Here, one equation is posed as a saddle point problem while the other equation acts as a constraint on the first by establishing V as a Lagrange multiplier. This is implemented by using divergence elements for the **D**-field and a discontinuous scalar Lagrange element for V. In this implementation, V is a lower polynomial order than the **D**-field. The extra degrees of freedom on the displacement field provide greater accuracy for this flux term. But as a result, the electric potential has a coarser representation though it is still accurate in the integral sense.

The **Force Calculation** feature and the **Electromechanics Multiphysics** coupling interface both use the **Maxwell Stress Tensor**, see Computing Electromagnetic Forces and Torques. When using the **D-V formulation**, this tensor is computed using the flux terms, **D** and **E** with this higher polynomial order.

A consequence of formulating the interface with the flux **D**-field as the dependent variable is that the types of boundary conditions are swapped. An essential boundary condition in the standard V-formulation is a natural boundary condition in the **D-V Formulation** and a natural boundary condition becomes an essential one. The features of the **Electrostatics** interface have been reformulated accordingly.

Effective Material Properties in Effective Media and Mixtures

One way of dealing with effective media or mixtures of solids in electromagnetic models is to replace them with a homogenized medium. The electric and magnetic properties of this medium are computed from the properties of each phase by means of an averaging formula.

There are several possible approaches to compute an average material property starting from the material properties and the volume fraction of each material.

The following sections illustrate the different formulas available to compute the *effective electric conductivity*, the *effective relative permittivity*, and the *effective relative permeability* of a homogenized medium. In the following, volume fractions of the materials are indicated with θ_i, where i is the material index, and they are assumed to be fractional (between 0 and 1). Up to five different materials can be specified as phases of the mixture. Typically, their volume fractions should add up to 1.

Effective Conductivity in Effective Media and Mixtures

Three methods are available to compute the averaged electric conductivity of the mixture.

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:

$$\sigma = \sum_{i=1}^{n} \theta_i \sigma_i = \theta_1 \sigma_1 + \theta_2 \sigma_2 + \dots$$

where σ_i is the conductivity of the material i. This is equivalent to a "parallel" system of resistivities.

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

$$\frac{1}{\sigma} = \sum_{i=0}^{n} \frac{\theta_i}{\sigma_i} = \frac{\theta_1}{\sigma_1} + \frac{\theta_2}{\sigma_2} + \dots$$

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:

$$\sigma = \prod_{i=0}^{n} \sigma_i^{\theta_i} = \sigma_1^{\theta_1} \sigma_2^{\theta_2} \dots$$

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 between these two.

Effective Relative Permittivity in Effective Media and Mixtures

Three methods are available to compute the averaged electric conductivity of the mixture.

VOLUME AVERAGE, PERMITTIVITY

If the relative permittivity of the two materials is not so different from each other, the effective relative permittivity ε_r is calculated by simple volume average:

$$\varepsilon = \sum_{i=1}^{n} \theta_i \varepsilon_i = \theta_1 \varepsilon_1 + \theta_2 \varepsilon_2 + \dots$$

where ε_i is the relative permeability of the material i.

If the permittivity is defined by second-order tensors (such as for anisotropic materials), the volume average is applied element by element.

VOLUME AVERAGE, RECIPROCAL PERMITTIVITY

The second method is the volume average of the inverse of the permittivities:

$$\frac{1}{\varepsilon} = \sum_{i=0}^{n} \frac{\theta_i}{\varepsilon_i} = \frac{\theta_1}{\varepsilon_1} + \frac{\theta_2}{\varepsilon_2} + \dots$$

If the permittivity is defined by a second-order tensor, the inverse of the tensor is used.

POWER LAW

A power law gives the following expression for the equivalent permittivity:

$$\varepsilon = \prod_{i=0}^{n} \varepsilon_i^{\theta_i} = \varepsilon_1^{\theta_1} \varepsilon_2^{\theta_2} \dots$$

The effective permittivity calculated by Volume Average, Permittivity is the upper bound, the effective permittivity calculated by Volume Average, Reciprocal Permittivity is the lower bound, and the Power Law average gives a value somewhere between these two.

Effective Relative Permeability in Effective Media and Mixtures

Three methods are available to compute the averaged electric conductivity of the mixture.

VOLUME AVERAGE, PERMEABILITY

If the relative permeability of the two materials is not so different from each other, the effective relative permeability μ_r is calculated by simple volume average:

$$\mu = \sum_{i=1}^{n} \theta_i \mu_i = \theta_1 \mu_1 + \theta_2 \mu_2 + \ldots$$

where μ_i is the relative permeability of the material i.

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

The second method is the volume average of the inverse of the permeabilities:

$$\frac{1}{\mu} = \sum_{i=0}^{n} \frac{\theta_i}{\mu_i} = \frac{\theta_1}{\mu_1} + \frac{\theta_2}{\mu_2} + \ldots$$

If the permeability is defined by a second-order tensor, the inverse of the tensor is used.

POWER LAW

A power law gives the following expression for the equivalent permeability:

$$\mu = \prod_{i=0}^{n} \mu_i^{\theta_i} = \mu_1^{\theta_1} \mu_2^{\theta_2} \ldots$$

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 between these two.

Archie's Law Theory

The electric 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 can have even lower conductivities. In variably saturated soils, the conductivity of air is roughly ten orders of magnitude lower than 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 nonconducting materials, Archie (Ref. 1) assumed that electric current are mainly caused by ion fluxes through 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 parameter that describes the connectivity of the pores. The cementation exponent normally varies between 1.3 and 2.5 for most sedimentary rocks and 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 $\varepsilon_r = 1$.

Reference for Archie's Law

1. G.E. Archie, "The Electric Resistivity as an Aid in Determining Some Reservoir Characteristics," *Trans. Am. Inst. Metal. Eng.*, vol. 146, pp. 54–62, 1942.

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 components. Electrolysis, conductors, and the computation of resistances of grounding plates are examples that involve conductive media with electric conductivities and electric currents. If you are uncertain of whether to use the Electric Currents or the Electrostatics interface, which both solve for the scalar electric potential V, refer to the section on Charge Relaxation Theory.

The Electric Currents Interface assumes that there is only one type of charge carrier. When there are multiple charge carriers, it is often necessary to solve the transport equation for each charge carrier as well as Poisson's equation with the Electrostatics interface.

In this section:

- Electric Currents Equations in Steady State
- Dynamic Electric Currents Equations
- Theory for Electrical Contact

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 electric 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:

$$-\nabla \cdot d(\sigma \nabla V - \mathbf{J}_e) = dQ_j \qquad (2\text{-}20)$$

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.

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

$$\nabla \cdot (\sigma \mathbf{E} + \mathbf{J}_e) + j\omega\rho = 0$$

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

$$\nabla \cdot (\sigma \mathbf{E} + \mathbf{J}_e) + \frac{\partial\rho}{\partial t} = 0$$

and generalized to handle current sources the resulting equation becomes:

$$-\nabla \cdot \frac{\partial}{\partial t}(\varepsilon_0 \nabla V - \mathbf{P}) - \nabla \cdot (\sigma \nabla V - \mathbf{J}_e) = Q_j \qquad (2\text{-}21)$$

In planar 2D the dynamic formulations also involves the thickness d in the z direction:

$$-\nabla \cdot d((\sigma + j\omega\varepsilon_0)\nabla V - (\mathbf{J}_e + j\omega\mathbf{P})) = dQ_j$$

$$-\nabla \cdot d\frac{\partial}{\partial t}((\varepsilon_0 \nabla V - \mathbf{P}) - \nabla \cdot d(\sigma \nabla V - \mathbf{J}_e)) = dQ_j$$

Theory for Electrical Contact

The Electrical Contact feature node has correlations to evaluate the joint conductance at two contacting surfaces.

The current densities at the upside and downside boundaries depend on the potential difference according to the relations:

$$\mathbf{n} \cdot \mathbf{J}_1 = -h_c(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = -h_c(V_2 - V_1)$$

At a microscopic level, contact is made at a finite number of spots as in Figure 2-1.
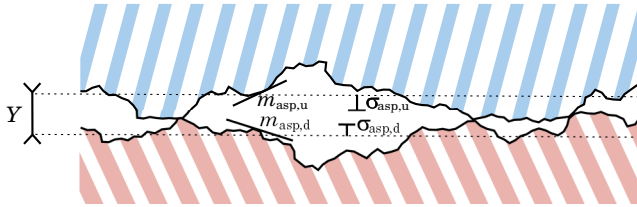
Figure 2-1: Contacting surfaces at the microscopic level.

SURFACE ASPERITIES

The microscopic surface asperities are characterized by the average height $\sigma_{u,\,asp}$ and $\sigma_{d,\,asp}$ and the average slope $m_{u,\,asp}$ and $m_{d,\,asp}$. The RMS values σ_{asp} and m_{asp} are (4.16 in Ref. 1):

$$\sigma_{asp} = \sqrt{\sigma_{u,\,asp}^2 + \sigma_{d,\,asp}^2} \qquad m_{asp} = \sqrt{m_{u,\,asp}^2 + m_{d,\,asp}^2}$$

CONSTRICTION CONDUCTANCE

Cooper–Mikic–Yovanovich (CMY) Correlation
The Cooper–Mikic–Yovanovich (CMY) correlation is valid for isotropic rough surfaces and has been formulated using a model assuming plastic deformation of the surface asperities. However, this model does not compute nor store the plastic deformations of the asperities. It means that, despite that a plastic deformation of the asperities is

assumed, this contact model has no memory. For example, if a load is applied twice the electrical contact is identical in both cases. The Cooper–Mikic–Yovanovich (CMY) correlation relates h_c to the asperities and pressure load at the contact interface:

$$h_c = 1.25\sigma_{contact}\frac{m_{asp}}{\sigma_{asp}}\left(\frac{p}{H_c}\right)^{0.95}$$

Here, H_c is the microhardness of the softer material, p is the contact pressure, and $\sigma_{contact}$ is the harmonic mean of the contacting surface conductivities:

$$\sigma_{contact} = \frac{2\sigma_u\sigma_d}{\sigma_u + \sigma_d}$$

📝 When σ_u (resp. σ_d) is not isotropic, it is replaced by its normal conductivity $\mathbf{n}^T\sigma_u\mathbf{n}$ (resp. $\mathbf{n}^T\sigma_d\mathbf{n}$).

The relative pressure p/H_c can be evaluated by specifying H_c directly or using the following relation (4.16.1 in Ref. 1) for the relative pressure using c_1 and c_2, the Vickers correlation coefficient and size index:

$$\frac{p}{H_c} = \left(\frac{p}{c_1\left(1.62\dfrac{\sigma_{asp}}{m_{asp}\sigma_0}\right)^{c_2}}\right)^{\frac{1}{(1+0.071c_2)}}$$

The coefficients c_1 and c_2 are the Vickers correlation coefficient and size index, respectively, and σ_0 is equal to 1 µm. For materials with Brinell hardness between 1.30 and 7.60 GPa, c_1 and c_2 are given by the correlation below (4.16.1 in Ref. 1):

$$\frac{c_1}{H_0} = 4.0 - 5.77\frac{H_B}{H_0} + 4.0\left(\frac{H_B}{H_0}\right)^2 - 0.61\left(\frac{H_B}{H_0}\right)^3$$

$$c_2 = -0.37 + 0.442\frac{H_B}{c_1}$$

The Brinell hardness is denoted by H_B, and H_0 is equal to 3.178 GPa.

Mikic Elastic Correlation

The Mikic correlation is valid for isotropic rough surfaces and assumes elastic deformations of surface asperities. It gives h_c by the following relation:

$$h_c = 1.54 \sigma_{\text{contact}} \frac{m_{\text{asp}}}{\sigma_{\text{asp}}} \left(\frac{\sqrt{2}p}{mE_{\text{contact}}} \right)^{0.94}$$

Here, E_{contact} is an effective Young's modulus for the contact interface, satisfying (4.16.3 in Ref. 1):

$$\frac{1}{E_{\text{contact}}} = \frac{1 - v_u^2}{E_u} + \frac{1 - v_d^2}{E_d}$$

where E_u and E_d are Young's moduli of the two contacting surfaces and v_u and v_d are Poisson's ratios.

⦵	Electrical Contact

REFERENCE

1. M.M. Yovanovich and E.E. Marotta, "Thermal Spreading and Contact Resistance," *Heat Transfer Handbook*, A. Bejan and A.D. Kraus, John Wiley & Sons, 2003.

Theory for the Electric Currents, Single Layer Shell Interface

The Electric Currents, Single Layer 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, Single Layer Shell interface is also available on boundaries in 2D geometries.

In this section:

- Electric Currents, Single Layer Shell Equations in Steady State
- Dynamic Electric Currents Equations

Electric Currents, Single Layer Shell Equations in Steady State

In the static study type, the physics 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 electric 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

$$-\nabla_t \cdot d\frac{\partial}{\partial t}((\varepsilon_0 \nabla_t V - \mathbf{P}) - \nabla_t \cdot d(\sigma \nabla_t V - \mathbf{J}_e)) = dQ_j$$

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

$$-\nabla \cdot (\mu_0 \nabla V_m - \mu_0 \mathbf{M}) = 0 \qquad (2\text{-}22)$$

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:

$$-\nabla \cdot d(\mu_0 \nabla V_m - \mu_0 \mathbf{M}) = 0$$

For a discussion about the boundary element method used with The Magnetic Fields, No Currents, Boundary Elements Interface, see Theory for the Boundary Elements PDE in the *COMSOL Multiphysics Reference Manual*.

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 Interface is used for 3D, 2D in-plane, and 2D axisymmetric components. Unless you have a license for the AC/DC Module, only 2D components involving out-of-plane currents and axisymmetric components involving azimuthal currents are supported.

In this section:

- Magnetostatics Equation
- Frequency Domain Equation
- Transient Equation
- Time Periodic Equation
- Maxwell's Equations (Quasistatic)
- Magnetic and Electric Potentials
- Gauge Transformations
- Selecting a Particular Gauge
- The Gauge and Equation of Continuity for Dynamic Fields
- Explicit Gauge Fixing/Divergence Constraint
- Ungauged Formulations and Current Conservation
- Time-Harmonic Magnetic Fields

Magnetostatics Equation

To derive the magnetostatic equation, start with Ampère's law for static cases $\nabla \times \mathbf{H} = \mathbf{J}$. The current is

$$\mathbf{J} \;=\; \sigma \mathbf{v} \times \mathbf{B} + \mathbf{J}_e$$

where $\mathbf{J}_e$ 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

$$\nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) = \mathbf{J}_e$$

which is the equation used in magnetostatics.

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

$$(j\omega\sigma - \omega^2\varepsilon_0)\mathbf{A} + \nabla \times (\mu_0^{-1}\nabla \times \mathbf{A} - \mathbf{M}) - \sigma\mathbf{v} \times (\nabla \times \mathbf{A}) = \mathbf{J}_e$$

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, illustrated here with the constitutive relation $\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$.

$$\sigma\frac{\partial \mathbf{A}}{\partial t} + \nabla \times (\mu_0^{-1}\nabla \times \mathbf{A} - \mathbf{M}) - \sigma\mathbf{v} \times (\nabla \times \mathbf{A}) = \mathbf{J}_e$$

The term involving the velocity only applies in the 2D and 2D axisymmetric formulations.

Time Periodic Equation

The time periodic equation this physics interface solves is Ampère's law, illustrated here in the 2D case with in-plane fields and assuming materials with a relative permeability μ_r.

$$2\pi\sigma f_{TP}\frac{\partial A_Z}{\partial \phi} + \mathbf{e}_Z \cdot \nabla \times (\mu^{-1}\nabla \times (A_Z\mathbf{e}_Z)) = J_Z\mathbf{e}_Z$$

The dependent variable A_Z is a function of the phase. The same may be true for the other variables, for example μ_r and the current density J_Z. This phase dependence may be intrinsic, or it may be coming from a definition containing other phase dependent variables. As an example, if the magnetic flux density $\mathbf{B}(\phi) = \nabla \times (A_Z(\phi)\mathbf{e}_Z)$ and its norm $B(\phi) = |\mathbf{B}(\phi)|$ are introduced, it is possible to express the relative permeability as

$$\mu_r(\phi) = \frac{B(\phi)}{\mu_0 H(B(\phi))}$$

in order to model a region where a B-H relation is used as input. Here, $H(B)$ is the B-H function, which relates the magnetic field to the magnetic flux density.

Maxwell's Equations (Quasistatic)

For cases where currents and electromagnetic fields vary slowly, the induced displacement current can be ignored ($\partial\mathbf{D}/\partial t = 0$). This assumption, referred to as the quasistatic approximation, is widely used in modeling low-frequencies electromagnetics where the dimensions of the structure is small compared to the wavelength. With this approximation, rewrite *Maxwell's equations* (Equation 2-1) in the following manner:

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

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

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

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

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

where $\mathbf{J}_e$ is an externally generated current density and $\mathbf{v}$ is the velocity of the conductor.

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

$$\sigma\frac{\partial \mathbf{A}}{\partial t} + \nabla \times (\mu_0^{-1}\nabla \times \mathbf{A} - \mathbf{M}) - \sigma\mathbf{v} \times (\nabla \times \mathbf{A}) + \sigma\nabla V = \mathbf{J}_e \qquad (2\text{-}23)$$

The equation of continuity, which is obtained by taking the divergence of the above equation, adds the following equation:

$$\nabla \cdot \left(-\sigma\frac{\partial \mathbf{A}}{\partial t} + \sigma\mathbf{v} \times (\nabla \times \mathbf{A}) - \sigma\nabla V + \mathbf{J}_e\right) = 0 \qquad (2\text{-}24)$$

Equation 2-23 and Equation 2-24 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

It is important to observe 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 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 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 electric 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 Gauge and Equation of Continuity for Dynamic Fields). 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 variable ψ 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 toward 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 unphysical 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 Explicit Gauge Fixing/Divergence Constraint.

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):

$$\nabla \times \mathbf{H} = \mathbf{J} = \sigma(\mathbf{E} + \mathbf{v} \times \mathbf{B}) + (j\omega \mathbf{D} + \mathbf{J}_e)$$

In the transient case, the inclusion of this term leads 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:

$$-\nabla \cdot ((j\omega\sigma - \omega^2\varepsilon_0)\mathbf{A} - \sigma\mathbf{v} \times (\nabla \times \mathbf{A}) + (\sigma + j\omega\varepsilon_0)\nabla V - (\mathbf{J}_e + j\omega\mathbf{P})) = 0$$

$$(j\omega\sigma - \omega^2\varepsilon_0)\mathbf{A} + \nabla \times (\mu_0^{-1}\nabla \times \mathbf{A} - \mathbf{M}) - \sigma\mathbf{v} \times (\nabla \times \mathbf{A}) + (\sigma + j\omega\varepsilon_0)\nabla V = \mathbf{J}_e + j\omega\mathbf{P}$$

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,

$$(j\omega\sigma - \omega^2\varepsilon_0)\tilde{\mathbf{A}} + \nabla \times (\mu_0^{-1}\nabla \times \tilde{\mathbf{A}} - \mathbf{M}) - \sigma\mathbf{v} \times (\nabla \times \tilde{\mathbf{A}}) = \mathbf{J}_e + j\omega\mathbf{P}$$

Working with $\tilde{\mathbf{A}}$ is often the best option when it is possible to specify all source currents as external currents $\mathbf{J}_e$ or as surface currents on boundaries.

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, and 2D axisymmetric components. For a deeper theoretical background to the magnetic vector potential and electric scalar potential used below, see the Theory for the Magnetic Fields Interface.

In this section:

- Stationary Equations
- Time Domain Equations
- Frequency Domain Equations

Stationary Equations

To derive the stationary 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

$$\mathbf{J} = \sigma \mathbf{v} \times \mathbf{B} - \sigma \nabla V + \mathbf{J}_e$$

where $\mathbf{J}_e$ 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

$$\nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + \sigma \nabla V = \mathbf{J}_e$$

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:

$$-\nabla \cdot (-\sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + \sigma \nabla V - \mathbf{J}_e) = 0$$

$$\nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + \sigma \nabla V = \mathbf{J}_e$$

Time Domain Equations

The time domain equations are based on Maxwell's Equations (Quasistatic). The time domain formulation is the same as the Stationary Equations, except that the electric field is defined as

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

The time domain formulation is inherently ungauged, see Gauge Transformations.

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

$$\nabla \times \mathbf{H} = \mathbf{J} = \sigma(\mathbf{E} + \mathbf{v} \times \mathbf{B}) + (j\omega \mathbf{D} + \mathbf{J}_e)$$

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

$$(j\omega\sigma - \omega^2 \varepsilon_0)\mathbf{A} + \nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + (\sigma + j\omega\varepsilon_0)\nabla V - j\omega \mathbf{P} = \mathbf{J}_e$$

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:

$$-\nabla \cdot ((j\omega\sigma - \omega^2 \varepsilon_0)\mathbf{A} - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + (\sigma + j\omega\varepsilon_0)\nabla V - (\mathbf{J}_e + j\omega \mathbf{P})) = 0$$

$$(j\omega\sigma - \omega^2 \varepsilon_0)\mathbf{A} + \nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + (\sigma + j\omega\varepsilon_0)\nabla V - j\omega \mathbf{P} = \mathbf{J}_e$$

Theory for the Magnetic Field Formulation Interface

The Magnetic Field Formulation Interface is used for 3D, 2D in-plane, and 2D axisymmetric components. The alternative formulation of the magnetic problem is particularly suitable to model materials having a highly nonlinear electric conductivity, such as superconductors. This section discusses the Equation System.

> 🔍 For a deeper theoretical background to the magnetic vector potential used, see Maxwell's Equations (Quasistatic).

Equation System

To derive the equation system in this physics interface, start with Faraday's law:

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

The electric field is computed from the current density, using an appropriate constitutive relation:

$$\mathbf{E} = \mathbf{E}_{\mathrm{m}}(\mathbf{J} - \mathbf{J}_e) + \mathbf{v} \times \mathbf{B}$$

where $\mathbf{J}_e$ is an externally generated current density, $\mathbf{v}$ is the relative velocity of the domain in the physics interface's or node's frame, and $\mathbf{E}_{\mathrm{m}}(\mathbf{J})$ is a (possibly nonlinear) relation sometimes called *E-J characteristic*, which describes the electric field in the local (material) frame as a function of the current density. The total current density J is finally obtained from Ampère's Law:

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

Combining these equations give the first equation in the system applied by the Magnetic Field Formulation interface:

$$\nabla \times (\mathbf{E}_{\mathrm{m}}(\mathbf{J} - \mathbf{J}_e) + \mathbf{v} \times \mathbf{B}) + \frac{\partial \mathbf{B}}{\partial t} = 0$$

This equation must be modified in different studies and according to the chosen constitutive relation. In stationary studies, the equation becomes:

$$\nabla \times (\mathbf{E}_{\mathrm{m}}(\mathbf{J} - \mathbf{J}_e) + \mathbf{v} \times \mathbf{B}) = 0$$

while in frequency domain is:

$$\nabla \times (\mathbf{E}_{\mathrm{m}}(\mathbf{J} - \mathbf{J}_e) + \mathbf{v} \times \mathbf{B}) + j\omega\mathbf{B} = 0$$

> The term involving the velocity only applies in the 2D and 2D axisymmetric formulations.

The second equation in the system is the magnetic Gauss' law:

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

This equation is added with two purposes:

- Make the problem numerically stable when using curl elements. The feature adding this equation can only be applied when the magnetic field is discretized using curl elements, that is, in 3D components, and 2D or 2D axisymmetric components when the in-plane components are solved for.

- In magnetostatics, the magnetic flux density $\mathbf{B}$ does not appear in the first equation. To allow modeling of magnetic materials and permanent magnets, it is necessary to include $\mathbf{B}$ (with its constitutive relation) through the Magnetic Gauss' Law node.

E-J CHARACTERISTIC FOR THE ELECTRIC FIELD COMPUTATION

The $\mathbf{E}_{\mathrm{m}}(\mathbf{J})$ characteristic introduced in the previous section can be specified in different ways in the Faraday's Law node. The most common case is a linear relation between $\mathbf{E}_{\mathrm{m}}$ and $\mathbf{J}$. In magnetostatics and in time-dependent studies (in the quasistatic approximation) this relation is specified by the resistivity or the electric conductivity:

$$\mathbf{E}_{\mathrm{m}} = \rho\mathbf{J}$$
$$\mathbf{E}_{\mathrm{m}} = \sigma^{-1}\mathbf{J}$$

σ and ρ can be tensor quantities, in which case the inverse is intended as the inverse of a tensor. In frequency-domain, the displacement current is included, giving the following relations:

$$\mathbf{E}_m = ((\rho)^{-1} + j\omega\varepsilon_0\varepsilon_r)^{-1}\mathbf{J}$$

$$\mathbf{E}_m = (\sigma + j\omega\varepsilon_0\varepsilon_r)^{-1}\mathbf{J}$$

If the E-J characteristic is nonlinear, such in the case of a superconductor, the function can be expressed explicitly or as a material property.

⫿	*Superconducting Wire*: Application Library path **ACDC_Module/ Devices,_Superconducting/superconducting_wire**

Theory for the Magnetic Fields, Currents Only Interface

In the area of microelectronics, it is of interest to compute the lumped inductance matrix of a group of coils or conductors. The geometry of the coils is usually 'open' and the modeling of a closed current loop is not of interest. In this case, the ability of modeling nondivergence-free currents is desired. As a complement to The Magnetic Fields Interface, which can only be used to model divergence-free currents, The Magnetic Fields, Currents Only Interface is designed to support both divergence-free and nondivergence-free currents. In free space it returns the value of the Biot–Savart integral.

Start with Ampère's law for static cases $\nabla \times \mathbf{H} = \mathbf{J}$ and assume a uniform permeability in free space ($\mathbf{B} = \mu_0 \mathbf{H}$), the magnetostatic problem reads

$$\nabla \times \mathbf{B} = \mu_0 \mathbf{J} \qquad (2\text{-}25)$$

Using the definitions of magnetic potential, $\mathbf{B} = \nabla \times \mathbf{A}$, rewrite Equation 2-25 as

$$\nabla \times \nabla \times \mathbf{A} = \mu_0 \mathbf{J} \qquad (2\text{-}26)$$

Applying the Coulomb gauge $\nabla \cdot \mathbf{A} = 0$ and vector calculus identity $\nabla \times \nabla \times \mathbf{A} = -\nabla^2 \mathbf{A} + \nabla(\nabla \cdot \mathbf{A})$, Equation 2-26 reduces to

$$-\nabla^2 \mathbf{A} = \mu_0 \mathbf{J} \qquad (2\text{-}27)$$

Equation 2-27 implies that The Magnetic Fields, Currents Only Interface is a div-grad formulation, different from The Magnetic Fields Interface which is a curl-curl formulation. Taking the divergence of both sides of Equation 2-27, it is noticed that the divergence of the current is not necessarily equal to zero. Therefore, Equation 2-27 is able to model open coils or conductors.

Different from The Magnetic Fields Interface, which uses Curl elements as shape functions in 3D, The Magnetic Fields, Currents Only Interface uses Lagrange elements as shape functions. In this way, The Magnetic Fields, Currents Only Interface is able to handle the continuity over nonconformal mesh elements.

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, Coenergy, 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, Coenergy, 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 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

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

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

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

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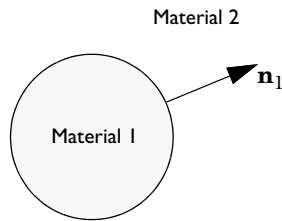
In certain cases, the stress tensor T can be divided into one part that depends on the electromagnetic field quantities and one part that is the mechanical stress tensor,

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

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 Solid Mechanics Interface in the *COMSOL Multiphysics Reference Manual.*

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 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 the divergence 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 Air is given by

$$\tau = \oint_{\partial\Omega_1} (\mathbf{r} - \mathbf{r}_0) \times (\mathbf{n}_1 T_2) dS$$

where $\mathbf{r}_0$ is a point on the axis of rotation. This follows from a derivation similar to the one made for forces.

Forces in Stationary Fields

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

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*: Application Library path **ACDC_Module/ Introductory_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.

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*: Application Library path **ACDC_Module/ Introductory_Electromagnetic_Forces/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 $-p\mathbf{I}$ 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 because 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

$$\tilde{\mathbf{E}} \ = \ \mathbf{E} + \mathbf{v} \times \mathbf{B} \qquad \text{(Electromotive intensity)}$$

$$\tilde{\mathbf{J}} \ = \ \mathbf{J} - \rho\mathbf{v} \qquad \text{(Free conduction current density)}$$

$$\tilde{\mathbf{P}} \ = \ \frac{\partial\mathbf{P}}{\partial t} + \mathbf{v}(\nabla \cdot \mathbf{P}) - \nabla \times (\mathbf{v} \times \mathbf{P}) \quad \text{(Polarization flux derivative)}$$

$$\tilde{\mathbf{M}} \ = \ \mathbf{M} + \mathbf{v} \times \mathbf{P} \qquad \text{(Lorentz magnetization)}$$

$$\tilde{\mathbf{H}} \ = \ \frac{\mathbf{B}}{\mu_0} - \varepsilon_0\mathbf{v} \times \mathbf{E} - \tilde{\mathbf{M}} \quad \text{(Magnetomotive intensity)}$$

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

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

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

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

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

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

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

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

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

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

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, Coenergy, and Virtual Work

Another technique to calculate forces is to derive the electromagnetic energy or coenergy 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 ENERGY, COENERGY, FORCE, AND TORQUE

For nonlinear magnetic materials without hysteresis, the magnetic energy and coenergy densities are, respectively,

$$dW_{\mathrm{m}} = \int_0^{\mathbf{B}} \mathbf{H} \cdot d\mathbf{B'}$$

$$dW'_{\mathrm{m}} = \int_0^{\mathbf{H}} \mathbf{B} \cdot d\mathbf{H'}$$

Knowing either of these, the other can be computed through the relation

$$dW_{\mathrm{m}} = \mathbf{H} \cdot \mathbf{B} - dW'_{\mathrm{m}}$$

For the case of linear materials, the magnetic energy and coenergy densities are equal.

The method of virtual work utilizes 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 and linear materials*, the total force and torque can be computed in the same way but with opposite signs,

$$\mathbf{F}_I = \nabla W_{\mathrm{m}}$$

$$T_I = \frac{\partial W_{\mathrm{m}}}{\partial \varphi}$$

Under the condition of *constant currents and nonlinear materials*, the total force and torque must be computed from the magnetic coenergy and with opposite signs,

$$\mathbf{F}_I = \nabla W'_{\mathrm{m}}$$

$$T_I = \frac{\partial W'_{\mathrm{m}}}{\partial \varphi}$$

ELECTRIC ENERGY, COENERGY, FORCE, AND TORQUE

For nonlinear dielectric materials without hysteresis, the electric energy and coenergy densities are, respectively,

$$dW_e = \int_0^{\mathbf{D}} \mathbf{E} \cdot d\mathbf{D'}$$

$$dW'_e = \int_0^{\mathbf{E}} \mathbf{D} \cdot d\mathbf{E'}$$

Knowing either of these, the other one can be computed through the relation

$$dW_e = \mathbf{E} \cdot \mathbf{D} - dW'_e$$

For the case of linear materials, the electric energy and coenergy densities are equal.

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 and linear materials*, the total electric force and torque on a system can be computed as

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

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

Under the condition of *constant potentials and nonlinear materials*, the total electric force and torque on a system must be computed from the electric coenergy as

$$\mathbf{F}_V = \nabla W'_\mathrm{e}$$

$$T_V = \frac{\partial W'_\mathrm{e}}{\partial \varphi}$$

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

See Deformed Geometry and Moving Mesh and Sensitivity Analysis in the *COMSOL Multiphysics Reference Manual*.

Electromagnetic Quantities

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

TABLE 2-2: 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)	-
Electric conductivity	σ	siemens/meter	S/m
Energy density	W	joule/meter^3	J/m^3
Force	$\mathbf{F}$	newton	N
Frequency	ν	hertz	Hz
Imaginary unit	j, i	(dimensionless)	-
Impedance	Z, η	ohm	Ω
Inductance	L	henry	H
Magnetic current density (surface)	$\mathbf{J}_{ms}$	volt/meter	-
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

TABLE 2-2: ELECTROMAGNETIC QUANTITIES.

QUANTITY	SYMBOL	SI UNIT	ABBREVIATION
Magnetic susceptibility	χ_m	(dimensionless)	-
Magnetization	$\mathbf{M}$	ampere/meter	A/m
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

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

2. J. Jin, *The Finite Element Method in Electromagnetics*, 2nd ed., Wiley-IEEE Press, 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," *Int. J. Num. Meth. Eng.*, vol. 19, no. 3, pp. 393–404, 1983.

7. G. Eriksson, "Efficient 3D simulation of thin conducting layers of arbitrary thickness," in 2007 *IEEE International Symposium on Electromagnetic Compatibility*, pp. 1–6. IEEE, 2007.

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. It also provides you an insight on how to approach the modeling of electromagnetics problems. A number of models available through the **Application Libraries** also illustrate the different aspects of the simulation process.

In this chapter:

- Preparing for Modeling
- Modeling Coils
- Computing Forces and Torques
- Computing Losses
- Computing Lumped Parameters
- Connecting to Electrical Circuits
- Part Libraries
- Material Libraries

Preparing for Modeling

This section is intended to guide you through some selection processes when modeling with the AC/DC Module. General modeling approaches and strategies are covered. Several topics about the art of modeling might not be found in ordinary textbooks on electromagnetic theory. This section discusses these topics:

- What Problems Can You Solve? — Can I use the quasistatic physics interfaces or do I need wave propagation?
- Selecting the Space Dimension for the Model Geometry — Is a 2D, 3D, or 2D 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?
- Meshing and Solving — What issues might arise with respect to meshing and solving?

Q	For general guidelines for effective modeling, see Building a COMSOL Multiphysics Model in the *COMSOL Multiphysics Reference Manual*.

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?

!	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 electromagnetics problems. See Table 1-1 in Overview of the User's Guide for a list of the preset study types available by physics interface.

Most of the time-dependent and time-harmonic formulations use a *quasistatic approximation*. The approximation assumes 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.

One major difference between quasistatic and high-frequency modeling is that the formulations depend on the *electrical size* of the structure. This dimensionless measure is the ratio of the largest distance between two points in the structure to the wavelength of the electromagnetic fields. The quasistatic physics interfaces in this module are suitable for simulations of structures with an electrical size in the range up to $1/10$.

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 quasistatic and the full Maxwell formulations can be used.

	Physics 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 **Application Libraries** 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 faster to solve. 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

> Ω 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.

▥	*Electromagnetic Forces on Parallel Current-Carrying Wires*: Application Library path **ACDC_Module/ Introductory_Electromagnetic_Forces/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).
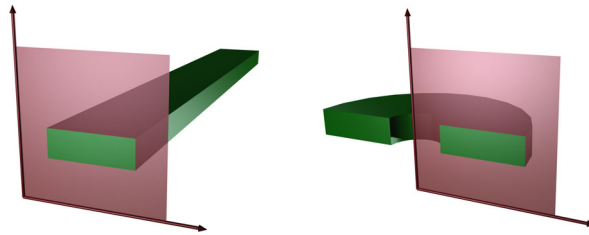
Figure 3-1: The cross sections and their real geometry for Cartesian coordinates (left) and cylindrical coordinates (axial symmetry).

Axial Symmetry (Cylindrical Coordinates)

	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. Since 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).

> *High Voltage Insulator*: Application Library path **ACDC_Module/ Devices,_Capacitive/high_voltage_insulator**

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.

3D PROBLEMS

> 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.

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).

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.

 Eddy Currents: Application Library path **ACDC_Module/ Introductory_Electromagnetics/eddy_currents**

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 quasistatic 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.

 One-Sided Magnet and Plate: Application Library path **ACDC_Module/ Introductory_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.

	Eddy Currents: Application Library path **ACDC_Module/ Introductory_Electromagnetics/eddy_currents**

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 be formulated as a stationary problem in the frequency domain with complex-valued variables. 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 the Fourier analysis, it applies to all periodic signals with the exception of nonlinear problems. Examples of

typical frequency domain simulations are quasistatic 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 interface. This physics 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 four predefined study sequences.

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

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

- Frequency-Stationary, One-Way Electromagnetic Heating
  - Time-harmonic magnetic fields
  - Stationary heat transfer

- Frequency-Transient, One-Way Electromagnetic Heating
  - Time-harmonic magnetic fields
  - Transient heat transfer

The difference between the first two and the last two study sequences is that in the Frequency-Stationary and the Frequency-Transient studies, the coupling between the electromagnetic and the heat transfer problem is bidirectional. The electromagnetic problem defines a heat source for the heat transfer problem and the material properties in the electromagnetic problem are temperature dependent. For the Frequency-Stationary, One-Way Electromagnetic Heating and Frequency-Transient, One-Way Electromagnetic Heating study sequences, it is assumed that the material properties for the electromagnetic problem is temperature independent, so a

Frequency Domain study is added for the electromagnetic problem followed by a Stationary or Time Dependent study for the heat transfer problem.

<table>
<tr><td>🔍</td><td>AC/DC Module Study Availability</td></tr>
</table>

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 mesh 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 might be required.

It is of great importance to make sure the formulation defined by the physics interface is compatible with the solver. For example, in the Electrostatics or the Electric Currents interface, a ground boundary condition is usually required to make the model be solved with a direct solver.

In the *COMSOL Multiphysics Reference Manual*:

- Meshing
- The Gauge
- Studies and Solvers

Modeling Coils

In this section:

- About the Coil Features
- About the Coil Name
- Coil Excitation
- Coil Groups
- Lumped Parameter Calculations
- Using Coils in 3D Models
- Automatic Coil Geometry Analysis
- Accurate Coil Voltage Calculation

🔍
- The Magnetic Fields Interface
- The Rotating Machinery, Magnetic Interface
- The Magnetic and Electric Fields Interface

About the Coil Features

The coil features can be used to simplify the setup of magnetostatics and low-frequency electromagnetic models. In many such applications, the magnetic field is generated by electric currents flowing in conductive materials (for example, cables, wires, coils, or solenoids). The coil features can be used to easily model these structures and to translate lumped quantities (currents and voltages) into distributed quantities (current densities and electric fields). This section describes the main Coil feature, which is available in The Magnetic Fields Interface, The Rotating Machinery, Magnetic Interface, and The Magnetic and Electric Fields Interface.

The Coil is available both as a domain and as a boundary feature. The latter can be used to model flat coils for which the thickness is negligible compared to the other two dimensions.

The **Coil** feature supports two different **Conductor model** options, which differ by the physical system represented and the modeling details.

- The **Single conductor** option models a single, solid region 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 option in a domain feature to model a single wire with a nonnegligible cross section; in a boundary feature it can be used for a thin conductive layer whose thickness is negligible (for example, in frequency domain, it is small with respect to the skin depth).

- The **Homogenized multiturn** option 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 conductor** model, 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. The boundary feature can be used when the wires are arranged in a thin region whose thickness is negligible. Note that capacitive coupling and skin effect are neglected in the homogenized model.

	For 2D and 2D axisymmetric geometries, the current applied by a Coil is always in the out-of-plane direction. Thus capacitive (in-plane) coupling is neglected.
	For 3D model geometries, the direction of the current flow is not easily determined. The coil domains have settings and subnodes to solve this problem. See Using Coils in 3D Models for more information.

	Selecting unconnected geometrical entities with a Coil connects them in parallel.

For examples of a **Single conductor** Coil, see:

- *Inductive Heating of a Copper Cylinder*: Application Library path
  ACDC_Module/Electromagnetic_Heating/inductive_heating

- *Modeling of a 3D Inductor* (described in detail in the *Introduction to the AC/DC Module* manual): Application Library path **ACDC_Module/Devices,_Inductive/inductor_3d**.

For examples of a **Homogenized multiturn** Coil see:

- *Magnetic Field of a Helmholtz Coil*: Application Library path
  ACDC_Module/Devices,_Inductive/helmholtz_coil

Inductor in an Amplifier Circuit: Application Library path
ACDC_Module/Electromagnetics_and_Circuits/inductor_in_circuit

WHAT MATERIAL TO USE FOR THE COIL?

The Coil domain feature is based on Ampère's Law and it requires material properties to set up the appropriate constitutive relations. The boundary feature also requires the electric conductivity when the Single conductor model is used. What is the correct choice of materials for the Coil domains or boundaries?

- When using the **Single conductor** option, the coil models a solid conductive domain (typically a metal). The material properties of the metal (electric permitting, magnetic permeability, and electric conductivity) should be used in this case.

- When using the **Homogenized multiturn** option, the conduction current (induced current) is assumed to flow only in the wires. To prevent induced current density from flowing in the domain, the region's electric conductivity σ is set to zero. The material used in the domain, only specifies the constitutive relations for the electric displacement field and the magnetic flux density. Therefore, choose or create a material that provides constitutive relations representative of the entire coil domain, not just of the material making up the metallic wires.

OTHER COIL FEATURES

Two coil features designed for special applications are available in the **Magnetic and Electric Fields** interface in 2D and 2D axisymmetric components:

- The RLC Coil Group feature, that models coils for which capacitive effects between coil turns and with other parts of the models are comparable with the inductive effects.

- The Single Conductor Coil feature, to introduce an externally applied current or voltage on domains where the Ampère's Law and Current Conservation feature is active. This feature is contributing, that is, it simply adds an external current density to the domain model represented by Ampère's Law and Current Conservation.

About the Coil Name

For all types of coil physics nodes, 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 nodes in a model.

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 *current excitation, voltage excitation* or, in certain cases, *power excitation*. 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.

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

Coil with Single Conductor model
When specifying a total current I_{coil}, the coil applies an external electric field in the direction of the current flow, computed as:

$$E_e = \frac{V}{L} \tag{3-1}$$

where L is equal to the physics interface thickness d for 2D models, $2\pi r$ for 2D axially symmetric models, and the coil length in 3D models. V is an unknown applied potential. The potential V is solved for using an additional equation, which constrains the total integrated current to be equal to the current value I_{coil} specified, acting as a controller.

Coil with Homogenized Multiturn Model
When specifying a total current I_{coil}, the coil applies an external current density in the direction of the wires as in Equation 3-2 where N is the number of turns which are specified and A is the total cross section area of the coil domain.

$$J_e = \frac{NI_{coil}}{A} \tag{3-2}$$

Note that in this case, the coil voltage is simply a derived variable. Its value is given by the wire resistance times the excitation current, plus the induced voltage originating from inductive effects. The latter is determined by integrating the electric field along the wire.

Coil Group Mode
When the Coil Group mode is activated, additional equations are added to the model to enforce the equality of the current flowing in each domain or domain group.

Single-Turn Coil (in Magnetic and Electric Fields)
When specifying a total current I_{coil}, the out-of-plane component of the current density is defined as:

$$J_e = \frac{\sigma V}{L} \tag{3-3}$$

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.

VOLTAGE EXCITATION

Coil with Single Conductor model
When specifying a total voltage V_{coil}, the externally applied electric field is computed as:

$$E_e = \frac{V_{coil}}{L} \tag{3-4}$$

where V_{coil} is the applied voltage specified, and L is equal to the physics interface's thickness d for 2D models, to $2\pi r$ for 2D axially symmetric models, and the coil length in 3D models.

Coil with Homogenized Multiturn model
When specifying a 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.

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

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

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

where L is equal to the physics interface's thickness d for 2D models, $2\pi r$ for 2D axially symmetric models, and the coil length in 3D models. The expression $\sigma_{coil} a_{coil}$ 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 Mode
When Coil Group mode is activated, additional conditions are added to the model to constrain the sum of all the voltages in the domains or domains groups to be equal to the specified voltage:

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

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.

Single-Turn Coil (in Magnetic and Electric Fields)
When specifying a total voltage V_{coil}, the coil applies an out-of-plane external current density computed as:

$$J_e = \frac{\sigma V_{coil}}{L} \tag{3-7}$$

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.

POWER EXCITATION

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

$$V_{coil} I_{coil} = P_{coil}$$

in stationary and time-dependent study steps, and

$$\frac{1}{2} \text{real}(V_{coil} I_{coil}^*) = P_{coil}$$

in frequency domain (cycle average). Select this option to specify the input power for the coils. The problem becomes nonlinear with these settings.

📝

- The **Power** option is not available for 3D models due to the increased complexity of the problem compared to 2D models.
- The **Power** option does not work with the Frequency Domain, Perturbation study.

❗

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

Be aware that, in general, the values of the voltage and the current might 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 Manual* for a possible solution to this problem.

Coil Groups

The Coil nodes, in 2D and 2D axisymmetry, present an additional **Coil Group** checkbox that allows to easily model one or more coil features connected in series. Each domain in the coil feature selection is considered a separate turn of the coil. Use this feature in 2D and 2D axisymmetric models for coils that cross the modeling plane more than once.

The Coil Group mode 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 base coil feature, 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.

Special subnodes can be added to the Coil nodes once the Coil Group checkbox has been selected in order to better control the setup of the connections:

Domain Group
When the Coil Group mode is active, each domain in the selection will be considered individually as a series-connected entity. The Domain Group subnode can be used to mark a group of domains which should be treated as one domain for the purpose of the coil set up. Multiple instances of this subnodes can be added to mark different groups. Domains that do not belong to a group are connected individually in series.

Reversed Current Direction
Use this subnode to mark a geometrical entity where the current flows in the opposite direction, that is antiparallel to the out-of-plane direction (z or azimuthal direction). This feature can be applied to any entity in the coil's selection, regardless of the group (if any) it belongs to.

📑 In COMSOL Multiphysics version 4.3a and earlier, a separate **Coil Group Domain** node was available. This is replaced in the current version by a **Single-Turn Coil** domain node with the **Coil Group** checkbox selected.

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 scope of the physics interface (for example, `mf.` for the Magnetic Fields interface) must be added to the beginning of the variable name — for example, `mf.ICoil_1` is the total coil current flowing in the coil 1 in the physics interface identified by the name `mf`. A brief description of each of the variables follows.

THE VOLTAGE, CURRENT, AND POWER VARIABLES

The coil nodes 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.

STATIONARY AND TIME DEPENDENT STUDIES

In Stationary and Time Dependent studies, the coil feature defines the coil resistance variable, R_{coil}, with the formula:

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

FREQUENCY DOMAIN STUDIES

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

TABLE 3-1: LUMPED VARIABLES FOR THE FREQUENCY DOMAIN STUDIES.

VARIABLE	EQUATION
Impedance	$Z_{\text{coil}} = \dfrac{V_{\text{coil}}}{I_{\text{coil}}}$
Resistance	$R_{\text{coil}} = \text{real}(Z_{\text{coil}})$
Inductance	$L_{\text{coil}} = \dfrac{\text{imag}(Z_{\text{coil}})}{\omega}$
Reactance	$X_{\text{coil}} = \text{imag}(Z_{\text{coil}})$
Admittance	$Y_{\text{coil}} = Z_{\text{coil}}^{-1}$

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:

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

🔍
- lindev in the *COMSOL Multiphysics Reference Manual*
- Coil
- Single Conductor Coil
- RLC Coil Group

The Single-Turn Coil Feature

The Single Conductor Coil feature models a single, solid region 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 the Single-Turn Coil domain node to model a single wire with a nonnegligible cross section; the Single-Turn Coil boundary node can be used for a thin conductive layer whose thickness is negligible (for example, in frequency domain, it is small with respect to the skin depth).

!
In COMSOL Multiphysics version 5.2 and earlier, the **Single-Turn Coil** was available in The Magnetic Fields Interface and The Rotating Machinery, Magnetic Interface. The functionality provided by this feature has been incorporated in the Coil feature in the form of the **Single conductor** option.

To preserve backward compatibility, the Single-Turn Coil features present in models created in version 5.2 and earlier will still be available when opening the models in the current version, but it is recommended to replace them with the new Coil feature.

💡
Selecting unconnected geometrical entities with a Single-Turn Coil connects them in parallel.

For examples of a Single-Turn Coil, see:

- *Inductive Heating of a Copper Cylinder*: Application Library path **ACDC_Module/Electromagnetic_Heating/inductive_heating**

- *Modeling of a 3D Inductor* (described in detail in the *Introduction to the AC/DC Module* manual): Application Library path **ACDC_Module/ Devices,_Inductive/inductor_3d**.

Using Coils in 3D Models

The Coil feature applies an external electric field or current density along the direction of the coil. The nodes require additional settings in 3D models to determine the geometry and the direction of the current flow.

COIL FEATURE

The options available in a Coil feature depend on the chosen **Conductor model**:

- When **Single conductor** is used, the coil direction is computed in the dedicated preprocessing study step **Coil Geometry Analysis**.

- When **Homogenized multiturn** is used, the **Coil type** input determines how the direction of the current flow is specified. The available alternatives are: **Linear**, **Circular**, **Numeric**, and **User defined**. The Numeric option computes the current flow in the dedicated preprocessing study step **Coil Geometry Analysis**. The other three options do not require any preprocessing.

- The **Single conductor Conductor model** and the **Homogenized multiturn Conductor model** support the **Stationary**, **Frequency Domain** and **Time Dependent** studies.

- For a detailed description about the usage of the **Conductor model** together with the **Coil Geometry Analysis** study step, see Automatic Coil Geometry Analysis.

In the **Homogenized multiturn** case, the coil type determines how the direction of the wires constituting the coil is specified, as well as the method used to compute the average length and the average cross-section area of the domain or boundary. The coil length and coil area are used to compute lumped variables, such as the induced voltage or the total resistance.

For models representing only a part of the geometry (due to symmetry), the total length and cross-section area of the coil (referred to as coil length and coil area) are computed by multiplying the domain or boundary length and area by the appropriate multiplication factors specified under **Symmetry specification** in the subfeatures.

The following sections detail the possible options available for the Coil feature.

Homogenized Multiturn Model — Linear Coil Type

In a *linear coil*, the wires are straight and parallel. To specify the direction of the wires, use the default Coil Geometry subnode to 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 (shown in the Graphics window with a red arrow), while the length of the wires in the domain is the total length of the edges. If the length of the edges does not correspond to the length of the domain, select the **Override length of the edges** checkbox and specify the correct length. For domain features, the average domain cross-section area is computed from the domain volume and the length. To specify another cross-section area, select the **Override domain area** checkbox and specify the correct area.

Homogenized Multiturn Model — Circular Coil Type

In a *circular coil*, the wires are wound in circles around a common axis. To specify the direction of the wires, use the default a Coil Geometry subnode to select a group of edges forming a circle or a part of a circle around the coil's axis. From the selected edges, the coil axis is computed and the direction of the wires is taken to be the azimuthal direction around the axis, as marked by the red arrows. The coil length used is computed as the coil volume divided by the coil cross sectional area, unless the **Use robust geometry analysis method** box is checked. When the robust method is used, the coil length is simply the length of the selected edges.

The geometry analysis algorithm can determine the axis and direction only if there is a geometry in the model. If the model does not have a geometry, for example when using imported meshes, use the alternative analysis method by selecting the **Use robust geometry analysis method** checkbox. This method can be used even without a geometry, but it requires that the selected edges form a complete circle to work correctly. If neither the default method nor the alternative method work, an alternative is to set the **Coil type** to **User defined** and manually specify the direction of the wires using a Cylindrical System.

Homogenized Multiturn Model — Numeric Coil Type

In a *numeric coil*, the path of the wires in the coil is computed numerically in an additional study step during the solution. This allows the modeling of coils with

complex shapes. To set up the numerical analysis computation of the current flow in a coil, additional information on the coil geometry must be provided by means of the default **Geometry Analysis** subnode and the boundary conditions under it.

By default, the Input (for Geometry Analysis) boundary condition is available under the **Geometry Analysis** subnode. For open coils (whose ends are on exterior boundaries), apply this condition on the input boundary, that is, the boundary at which the wires enter the coil domain. Right-click the Geometry Analysis node, add an Output (for Geometry Analysis) subnode, and apply it on the exterior boundary where the wires exit the domain.
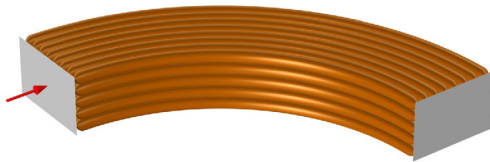
For closed-loop coils, apply the Input boundary condition on an interior boundary orthogonal to the direction of the wires.
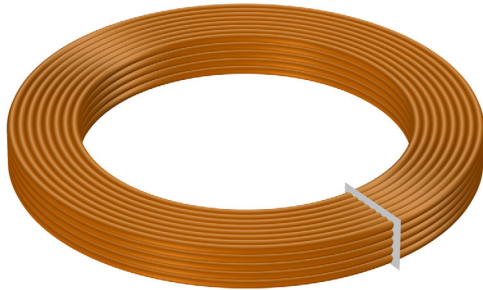
To complete the setup, add a **Coil Geometry Analysis** study step to the study, before the main study step.

!	The Coil Geometry Analysis can process multiple coils in the same step. In order to analyze only certain coils, select the **Solve only specific coils** checkbox and enter the coil names as a comma-separated list.

Homogenized Multiturn Model — User-Defined Coil Type

In a *user-defined* coil, the current flow (the direction of the wires) can be entered as an arbitrary vector field in the **User-Defined Coil Geometry** subnode. The vector field entered can be an analytical expression or the solution of another physics (for instance, the vector field computed by a **Curvilinear Coordinates** interface). In order for the magnetic problem to have a solution, the vector field must be as much as possible divergence-free, meaning that the current flow cannot have sources or sinks within the simulation domain nor can it originate from interior boundaries. In practice, this means that the current flow must either be terminated on exterior boundaries, or it must be closed in a loop.

Since the coil feature does not have any information on the geometry, additional settings are required:

- Enter the **Domain length** (for domain features) or the **Boundary length** (for boundary feature), which should be the average length of the coil from the input to the output — the average length of a wire, not accounting for symmetry nor the number of turns.

- For domain features, use the subnodes to **User Defined Coil Geometry** — **Input** and **Output** — to select the coil input and output boundaries, where the current flow enters or leaves the simulation domain. If the coil is closed in a loop, no selection is required.

Single Conductor Model

The setup of the Coil when using the Single conductor model is similar to the setup required for the Homogenized Multiturn Model — Numeric Coil Type case, the only difference being that the Coil Geometry Analysis study step will compute the physical current flow, instead of the direction of the wires. Refer to that section for more information.

SINGLE-TURN COIL

The 3D Single Conductor Coil feature was available in The Magnetic Fields Interface in previous versions of COMSOL Multiphysics. This feature is obsolete — the recommended approach to model the same physical system is to use a **Coil** feature with the **Single conductor** model, or The Magnetic and Electric Fields Interface.

The feature is still available for backward compatibility when opening models created in previous versions of the software.

This node represents a solid conducting domain, typically a wire or a coil, with a nonnegligible cross section. The boundary node represents a conducting thin layer whose thickness is small (also compared to the skin depth). 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 node, it is possible to specify the material properties that are used in the continuity equation.

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

!	To ensure that the current continuity equation has a physical solution, terminate a coil domain on external boundaries only, or close it into a loop.

A Gap Feed subnode 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 interior boundaries to the conductive domain and is useful for modeling closed loops.

!	For the Single-Turn Coil 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.

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

Automatic Coil Geometry Analysis

In 3D models, it is possible to perform an automatic analysis of the geometry of a Coil to determine the local direction of the current flow in the domain given the input and output boundaries of the coil (or an interior boundary in the case of closed-loop coils). The analysis is performed in a dedicated study step, **Coil Geometry Analysis**, which is typically solved before the main study step.

The study step can be used to compute both the current flow when using the **Single conductor** model and the direction of the wires when using the **Homogenized multiturn** model.

In the case of a **Single conductor** coil, the study step solves a simple current conservation problem to determine the flow of a current applied to the metallic domain. In this case, the **Coil Geometry Analysis** uses the initial values for the conductivity. If the **Coil Geometry Analysis** is followed by a study step that alters the conductivity, for instance, the conductivity depends on the temperature, the path that the currents take may deviate from what the **Coil Geometry Analysis** initially expected. This may cause the direction of the electric fields used to excite the coil, and the direction used for integrating the induced currents to become an approximation, rather than being entirely consistent with the solution. In many cases this is a good approximation since most conductors used in electromagnetic devices offer a single, straightforward conduction path; and good electrical conductors are typically good thermal conductors as well. Therefore, the temperature gradients inside the conductor are small (note that a uniform change in conductivity will not affect the current direction). However, in 3D, when the temperature gradients (and the resulting gradients in conductivity) are large enough to cause a significant shift in the direction of the currents, for example, when modeling sensors like thermistors, you can use the **Magnetic and Electric Fields** interface instead. In case of doubt, use the **Magnetic and Electric Fields** interface as a means of validation.

In the case of a **Homogenized multiturn** coil, the feature computes a vector field **e** which represents the local wire density in the coil, as well as the length and average cross-section of the wires. Note that the vector field **e** set by the **Coil Geometry Analysis** will remain valid in the following study steps since the current direction is assumed to be dictated by the direction of the wire bundle, not the local conductivity, which is different from a **Single conductor** coil. The vector variable eCoil can be plotted (for

example, in a **Streamline** or **Arrow Volume** plot) to visualize the computed direction of the wires.

	The **Coil Geometry Analysis** study step must precede the main study step (for example, a Stationary study step) in which the Coil is used.

	The **Coil Geometry Analysis** study step analyzes simultaneously the geometry of all coils in the physics that are active in the study — even if they use different Conductor models. To analyze only specific coils, select the **Solve only specific coils** checkbox in the study step and enter the names of the desired coils in the **Coil names** input field as a comma-separated list.

The coil geometry analysis method tries to construct a coil wires density that is plausible for an actual bundle of thin wires. The computed wire density is constant over arbitrary cross section, without violating current conservation for the current flowing in the wires. The density will depend on the local cross-section area of the coil (for example, it will increase in choke points), and will remain approximately constant in bends and curved regions as long as the cross-section area does not change. The analysis method will compute a solution even in presence of sharp bends, but the resultant wire density may not be constant. For best results, try to avoid sharp bends in the coil geometry.

BACKWARD COMPATIBILITY WITH COIL CURRENT CALCULATION

The **Coil Geometry Analysis** replaces the **Coil Current Calculation** functionality available in previous versions of COMSOL Multiphysics. The new formulation is more robust and has a larger applicability than the previous one, as well as a number of improvements, such as support for solving multiple coils at once or for coils with nonconstant cross section. When opening a model saved in an older version of COMSOL Multiphysics, the Coil Current Calculation study steps are transparently updated to use the new formulation. It may however be necessary to regenerate the default solvers before proceeding with the solution.

ADVANCED SOLVER SETTINGS

Depending on the type of coils used in the model, the Coil Geometry Analysis may use a Stationary solver with a Segregated node to perform the analysis. The default solver settings should suffice in most cases, but they can be adjusted if the analysis fails to provide a satisfying solution. The Number of iterations in the Segregated solver can be

used to tune the robustness of the method. Lower values improve the stability of the solution, but the condition that the wire density must be constant is weakened.

The Segregated solver uses Geometric Multigrid to solve the linear system associated with the analysis. If the Geometric Multigrid causes problems during the solution (for example, in presence of very coarse meshes), the direct solver can be used instead. Change the **Linear solver** in all Segregated Steps nodes to **Direct**.

Q	• About the Coil Features
	• Studies and Solvers in the *COMSOL Multiphysics Reference Manual*

Accurate Coil Voltage Calculation

This functionality improves the accuracy of the computation of the lumped coil voltage in time dependent and frequency domain studies, in 3D, when using the **Homogenized multiturn** conductor model. Other lumped parameters computed from the voltage using Ohm's Law, such as the coil impedance, inductance, and power, also benefit from the improved accuracy. The functionality is enabled by default in time dependent and frequency domain studies, and can be manually controlled using the corresponding checkbox in the Coil **Settings** window, **Stabilization** section. The **Stabilization** section can be shown by clicking the **Show More Options** button (👁) and selecting **Stabilization** in the **Show More Options** dialog. It is only available when the **Conductor model** is set to **Homogenized multiturn**.

For domain coils[1], the Accurate coil voltage calculation functionality performs a filtering of the external current density applied by the coil to obtain a current density vector field that is numerically divergence free. The filtering is performed by introducing an additional scalar degree of freedom V_f (the filtering variable) and solving the equation

$$\nabla \cdot (\nabla V_f - \mathbf{J}_u) = 0$$

in the coil domain, where $\mathbf{J}_u$ is the unfiltered external current density applied by the Coil feature. The filtered current density

$$\mathbf{J}_e = \nabla V_f - \mathbf{J}_u$$

---

1. For boundary coils, the filtering turns into a local in-plane 2D formulation. Apart from that, it is mathematically equivalent to the domain coil case.

is then applied as right-hand side in the magnetic equations. This filtering reduces the spurious electric fields in the solution, in turn improving the accuracy of the induced voltage (which is computed by integrating the electric field in the coil domain).

The filtering equation requires appropriate boundary conditions for the filtering variable, in particular, on the inlet and outlet boundaries of the coil, where the external current density is orthogonal. The approach used to set up these boundary conditions vary according to the coil geometry:

- For **Linear** and **Circular** coils, the geometry analysis algorithm automatically detects the inlet/outlet boundaries from the geometry. If this detection fails, use the Override Coil Inlet/Outlet subnode to manually select the inlet/outlet boundaries. Right-click the Coil node to add this subnode.

- For **Numeric** coils, the subnodes of the Geometry Analysis node are used to identify the inlet/outlet boundaries.

- For **User defined** coils, use the subfeatures of the User Defined Coil Geometry subnode — **Input** and **Output** — to select the coil inlet/outlet boundaries.

Computing Forces and Torques

In this section:

- Computing Electromagnetic Forces and Torques
- Model Examples — Electromagnetic Forces

Computing Electromagnetic Forces and Torques

Two methods are available to compute 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

In most AC/DC interfaces, the **Force Calculation** feature can be used to compute forces and torques using Maxwell stress tensor. In electric field and current interfaces, the force is calculated by integrating

$$\mathbf{n}_1 T_2 = -\frac{1}{2}\mathbf{n}_1(\mathbf{E} \cdot \mathbf{D}) + (\mathbf{n}_1 \cdot \mathbf{E})\mathbf{D}^T \tag{3-8}$$

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

In magnetic field interfaces, the expression

$$\mathbf{n}_1 T_2 = -\frac{1}{2}\mathbf{n}_1(\mathbf{H} \cdot \mathbf{B}) + (\mathbf{n}_1 \cdot \mathbf{H})\mathbf{B}^T \tag{3-9}$$

is integrated on the surface to obtain the force.

In the **Magnetic and Electric Fields** interface, both expressions (Equation 3-8 and Equation 3-9) 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 (Material 1).

With the Maxwell stress tensor, the force is expressed as

$$\mathbf{F} = \oint_{\partial\Omega_1} \mathbf{n}_1 T_2 dS \tag{3-10}$$

where $\partial\Omega$ represents the domain boundary. The torque is computed as

$$\tau = \oint_{\partial\Omega_1} (\mathbf{r} - \mathbf{r}_0) \times (\mathbf{n}_1 T_2) dS \qquad (3\text{-}11)$$

where $\mathbf{r}_0$ is a point on the axis of rotation. 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 `<name>.FLtz<component>` are available both in domains and on boundaries (in the case of surface currents).

!	For moving objects containing magnetic materials such as a magnetizable armature, the Lorentz force is no longer the only force contribution. In this case, the force contribution from the magnetic polarization might need to be considered. This can be done by adding the **Force Calculation** feature which computes the force using Maxwell stress tensor. Note that the **Force Calculation** feature includes the force contribution not only from magnetic polarization but also from the Lorentz force, that is, the total electromagnetic force.

CYCLE-AVERAGED QUANTITIES

In Frequency Domain studies, the formulas presented above can be used to compute the instantaneous force and torque by taking the real part of all the variables involved. It is often more interesting to compute the average of these quantities over a cycle to, for example, couple the electromagnetic forces to another physics. The cycle-averaged Maxwell's stress tensor (including both electric and magnetic forces) is computed as:

$$\mathbf{n}_1 T_{\text{av}, 2} = \frac{1}{2} Re\left(-\frac{1}{2}\mathbf{n}_1(\mathbf{E}^* \cdot \mathbf{D}) + (\mathbf{n}_1 \cdot \mathbf{E}^*)\mathbf{D}^T - \frac{1}{2}\mathbf{n}_1(\mathbf{H}^* \cdot \mathbf{B}) + (\mathbf{n}_1 \cdot \mathbf{H}^*)\mathbf{B}^T\right)$$

ARKKIO'S METHOD

In the **Rotating Machinery, Magnetic** interface, in addition to the **Force Calculation** feature, the **Arkkio Torque Calculation** feature based on Arkkio's method is available. Arkkio's method is a variant of the Maxwell stress tensor method, specifically useful in modeling rotating machines where the thin air gap separates rotating parts. In this

case, the normal $\mathbf{n}_1$ of the air gap can be defined as the radial outward unit vector. Arkkio's method computes the torque as

$$\tau = n_{\text{sec}} \oint_\Omega \frac{F_\phi r_A}{r_{\text{out}} - r_{\text{in}}} dV$$

$$F_\phi = \frac{B_r B_\phi}{\mu_0}$$

$$\Omega:\{r_{\text{in}} < r_A < r_{\text{out}}\}$$

(3-12)

where F_ϕ is the azimuthal force density; B_r, B_ϕ are the radial and azimuthal magnetic flux density, respectively; r_A is the distance from the rotation center; n_{sec} is the number of sectors. Different from surface integral Equation 3-11 used in Maxwell stress tensor method, the torque expression for Arkkio's method (Equation 3-12) is a volume integral on the domain Ω, yielding a less mesh-dependent result.

Model Examples — Electromagnetic Forces

There are a number of examples in the AC/DC Module Application 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.

> 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 a nonsolid medium (or when this is a good approximation).

- *Electromagnetic Forces on Parallel Current-Carrying Wires*: Application Library path **ACDC_Module/ Introductory_Electromagnetic_Forces/parallel_wires**
- *Permanent Magnet*: Application Library path **ACDC_Module/ Introductory_Magnetostatics/permanent_magnet**

Computing Losses

Computing electromagnetic losses is of significant importance in electromagnetics modeling as the losses can generate heat that is either useful or harmful. This section is devoted to the presentation of an overview of how low-frequency electromagnetic losses are modeled with the AC/DC Module.

In this section:

- Modeling Electromagnetic Losses
- Modeling Losses in the Frequency Domain
- Model Examples — Electromagnetic Losses

Modeling Electromagnetic Losses

In general, AC/DC interfaces compute electromagnetic losses by default. The volumetric loss density is made available as a global scalar variable with name `<name>.Qh`, where `<name>` is the **Name** of the physics interface. Depending on the interface and study type, the expression for computing electromagnetic losses can be different. Typical application scenarios and modeling approaches are:

- Use The Joule Heating Interface ( ) to model resistive heating and heating due to dielectric losses. This multiphysics interface adds an **Electric Currents** interface and a **Heat Transfer in Solids** interface. The multiphysics couplings add the electromagnetic power dissipation as a heat source, and the electromagnetic material properties can depend on the temperature. Different dielectric and conduction models are available in the **Electric Currents** interface.

- Use The Induction Heating Interface ( ) to model induction and eddy current heating. This multiphysics interface is based on the assumption that the magnetic cycle time is short compared to the thermal time scale (adiabatic assumption). Use this interface together with preset study types such as **Frequency-Transient** to model a variety of electromagnetic heating problems.

- Use the Loss Calculation subfeature under the **Ampère's Law** or domain **Coil** or **Faraday's Law** or **Magnetic Flux Conservation** feature to compute the copper loss due to the resistive heating or the iron loss due to the Hysteresis, eddy current and other effects. The feature is available with both Time Dependent and Frequency Domain studies. For Time Dependent studies, the loss calculation has to be used in combination with the **Time to Frequency Losses** study step.

DIELECTRIC LOSSES

For conductors with high conductivity, the main loss is the ohmic loss. For low-conductivity materials such as insulators, the dielectric losses are usually modeled in the frequency domain. Difference dielectric loss models are available in the **Constitutive Relation D-E** section of the interface. The frequency domain equations model dielectric losses with the *complex permittivity*, which is expressed as

$$\varepsilon_c = \varepsilon_0(\varepsilon' - j\varepsilon'')$$

where ε' is the real part of ε_r, and all losses (dielectric and conduction losses) are given by ε''. The dielectric loss model can also single out the losses from finite conductivity (so that ε'' only represents dielectric losses) resulting in:

$$\varepsilon_c = \varepsilon_0\left(\varepsilon' - j\left(\frac{\sigma}{\omega\varepsilon_0} + \varepsilon''\right)\right)$$

The complex permittivity can also be introduced as a loss tangent:

$$\varepsilon_c = \varepsilon_0\varepsilon'(1 - j\tan\delta)$$

where $\tan\delta$ is called the loss tangent.

When specifying losses through a complex permittivity or a loss tangent, conductivity is not allowed as an input (set to zero).

Depending on the Sign Convention, material data taken from the literature might have to be conjugated before using it in a model.

MAGNETIC LOSSES

The frequency domain equations allow for magnetic losses to be introduced as a *complex relative permeability*.

$$\mu_r = (\mu' - j\mu'')$$

where μ' is the real part of μ_r, and all magnetic losses are given by the imaginary part μ''.

The complex relative permeability model is available as one of the Magnetization models in the **Constitutive Relation B-H** section.

> Even though the electromagnetic losses are sometimes modeled in the frequency domain, it does not prevent you from performing a stationary or time dependent study later. Instead, there are many preset study types that can be used in different situations. Read this blog post for more information: https://www.comsol.com/blogs/which-study-type-should-i-use-for-my-electrothermal-analysis/

Model Examples — Electromagnetic Losses

There are a number of examples in the AC/DC Module Application Library showing how to compute electromagnetic losses in different situations.

The *Electrical Heating in a Busbar with Terminals* model demonstrates the usage of current terminal to compute the resistive heating of a busbar.

The *Inductive Heating of a Copper Cylinder* model demonstrates how to compute the inductive heating caused by eddy current and then use it as a heating source in a transient study of heat transfer.

The *Permanent Magnet Motor in 3D* model demonstrates how to compute electromagnetic losses in the magnets of a motor with the Time to Frequency Losses study.

> - *Electrical Heating in a Busbar with Terminals*: Application Library path **ACDC_Module/Introductory_Electric_Currents/busbar_terminal**
> - *Inductive Heating of a Copper Cylinder*: Application Library path **ACDC_Module/Electromagnetic_Heating/inductive_heating**
> - *Permanent Magnet Motor in 3D*: Application Library path **ACDC_Module/Devices,_Motors_and_Generators/pm_motor_3d**

Computing 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. This section will also discuss lumped ports and S-parameters.

In this section:

- Calculating Lumped Parameters with Ohm's Law
- Calculating Lumped Parameters Using the Energy Method
- Studying Lumped Parameters
- Lumped Parameter Conversion
- Stationary Source Sweep
- About Lumped Ports
- Lumped Port Parameters
- S-Parameters in Terms of Voltage and Electric Field
- S-Parameter Calculations
- S-Parameter Variables
- Port Sweeps/Manual Terminal Sweeps and Touchstone Export

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 3-2.
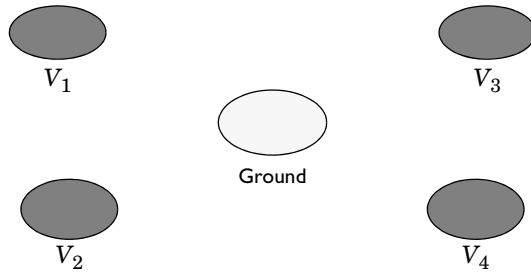
Figure 3-2: A five-electrode system with 4 terminals and one ground electrode.

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, $\mathbf{Y}$. 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 $\mathbf{Y}$.

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 or domain condition for each electrode. This 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 (only available as boundary condition and only 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.
- Magnetic Fields, Currents Only. Use the preset Stationary Source Sweep with Initialization study to compute lumped inductance and resistance matrices.

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.

In the main node of the physics 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, $\mathbf{Z}$, or the admittance matrix, $\mathbf{Y}$, 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 $\mathbf{L}$ is the inductance, $\mathbf{C}$ is the capacitance, $\mathbf{R}$ is the resistance, and $\mathbf{G}$ is the conductance. $\mathbf{S}$ is the S-parameter. The relations also include the following matrices

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

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

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

where Z_0 is the characteristic impedance.

For capacitance matrix calculations special transformations are available to obtain the mutual or SPICE capacitance matrix from the basic output from the solver. The latter is, for voltage excitation the so called Maxwell capacitance matrix and, for charge excitation the inverse Maxwell capacitance matrix. The Maxwell capacitance values, terminal charges, and terminal voltages are linked by the following matrix relation:

$$\begin{bmatrix} Q_1 \\ Q_2 \\ Q_3 \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} \\ C_{21} & C_{22} & C_{23} \\ C_{31} & C_{32} & C_{33} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \end{bmatrix} \tag{3-13}$$

where Q is charge, V is voltage, and C is the Maxwell capacitance matrix.

In order to analyze the response of a device, it is more common to use the form of a mutual capacitance matrix. The mutual capacitance matrix can be converted from the Maxwell capacitance matrix:

$$\begin{bmatrix} C_{m11} & C_{m12} & C_{m13} \\ C_{m21} & C_{m22} & C_{m23} \\ C_{m31} & C_{m32} & C_{m33} \end{bmatrix} = \tag{3-14}$$

$$\begin{bmatrix} C_{M11} + C_{M12} + C_{M13} & -C_{M12} & -C_{M13} \\ -C_{M21} & C_{M21} + C_{M22} + C_{M23} & -C_{M23} \\ -C_{M31} & -C_{M32} & C_{M31} + C_{M32} + C_{M33} \end{bmatrix}$$

where C_{Mij} are elements of the Maxwell capacitance matrix and C_{mij} are elements of the mutual capacitance matrix.

The basic steps of this conversion are:

1 Set each diagonal element of the mutual capacitance matrix to the sum of the corresponding row elements of the Maxwell capacitance matrix (that is, to get C_{m11}, sum C_{M1j} for all j).

2 Reverse the sign of all of the off-diagonal elements in the Maxwell capacitance matrix to get the corresponding mutual capacitance matrix elements.

This conversion can also be used to transform resistance and conductance matrices obtained from the **Electric Currents** physics. In that context the output resistance matrix corresponds to the inverse Maxwell capacitance matrix, electrode currents correspond to charges and the mutual or SPICE conductance matrix corresponds to the mutual or SPICE capacitance matrix.

You can compute conversions between the various matrices ($\mathbf{Z}$, $\mathbf{Y}$, $\mathbf{S}$, $\mathbf{C}_{\text{Max}}$, $\mathbf{C}_{\text{mut}}$) in a results table using the **Settings** window for the **Global Matrix Evaluation** node, which you can add under **Results** > **Derived Values**.

⦾	Global Matrix Evaluation in the *COMSOL Multiphysics Reference Manual*

Stationary Source Sweep

For the Electrostatics, the Electrostatics, Boundary Elements and the Electric Currents interfaces there is a special study step for efficient calculation of capacitance and resistance matrices respectively. The **Stationary Source Sweep** ($\lceil$ RLC) identifies all terminals (domain or boundary) in the model and excites them one at a time using a fixed charge or current (overriding the settings on the **Terminal** features). While one Terminal is excited, the others will be floating with zero net charge/current.

!	To get a proper matrix definition, at least one **Ground** node should be specified in the model. This is particularly important as the Stationary Source Sweep does not add any constraint on the electric potential.

Matrix Output
The resulting potential values at the terminals are used to extract one of the lumped matrices according to the following table.

PHYSICS	BASE LUMPED PARAMETER MATRIX
Electrostatics & Electrostatics, Boundary Elements	Inverse Maxwell capacitance matrix
Electric Currents	Resistance matrix

During postprocessing, other matrices can be produced via the **Global Matrix Evaluation**. The settings for Global Matrix Evaluation has a **Transformation** section that

allows for the output of the Maxwell capacitance matrix or the mutual capacitance matrix.

> | ! | If the **Generate default plots** checkbox on the Study node is enabled, default **Global Matrix Evaluation** nodes for matrix output are generated.

Solver Selection
The Stationary Source Sweep is optimized for solution speed. It works best with a direct solver as the LU decomposition has to be performed only once, followed by back substitution for the multiple source excitations (multiple right-hand sides). For 3D models, the Stationary Source Sweep generates both an iterative and a direct solver. The former will be selected if the model is having a large number of elements; that is, a number that is above a built-in threshold. For hardware with sufficiently large RAM, it is recommended to manually switch to the direct solver.

> | ! | For 3D models involving the **Electrostatics, Boundary Elements** physics, direct solvers scale poorly with problem size so some caution is recommended.

Voltage Output
With unit charge feeding the absolute value of the electric potential is difficult to predict. For this reason there is an auxiliary global variable — for example, `es.VexcTerm` — that represents the value of potential of the excited terminal. The default potential plots will typically show the electric potential scaled by this value.

Compatibility with Other Study Types
Combining the Stationary Source Sweep with the **Parametric Sweep** study node should only be done after careful consideration. If you need to perform more general parameter sweeps, it is often more appropriate to activate the manual terminal sweep in the settings of the main physics node and set up Parametric Sweeps manually. Other study nodes will work with Stationary Source Sweep as it was a standard Stationary (parametric) study. In particular, this is valid also for Optimization and Sensitivity studies.

About Lumped Ports

Lumped ports are used to model a connecting transmission line or as a voltage or current source applied between electrodes. For example, apply a lumped 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

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

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

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.

⊘	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.

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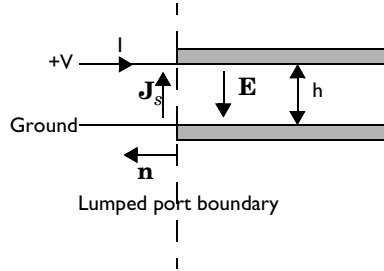
The transmission line current can be represented with a surface current at the lumped port boundary directed opposite to the electric field.

The impedance of a transmission line is defined as

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

and in analogy to this an equivalent surface impedance is defined 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.

> 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 in Terms of Voltage and 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/feeds, 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

The AC/DC interfaces have built-in support for S-parameter calculations. For connecting transmission lines and other lumped feeds, use a *lumped port* that approximates a connecting transmission lines or a voltage source with a known internal impedance. The lumped port should only be used when the port width is much smaller than the wavelength. In the Electric Currents and Electrostatics interfaces, use the *terminal* feature with the *terminated* setting to approximate a connecting transmission line or a voltage source with a known internal impedance. For a terminal the voltage measurement is always with respect to ground so at least one *ground* feature is also required in the model.

⚲	• For more details about lumped ports, see About Lumped Ports. • See Lumped Port for instructions to set up a model.

S-Parameter Variables

This 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:

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

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

In addition to the S-parameter variables, variables are also added for , as shown in the table below:

In the table above, is the tag for the physics interface. This tag can be different for different physics interface instances. N is the name of the port (a number).

Port Sweeps/Manual Terminal Sweeps and Touchstone Export

The Port Sweep Settings in the Magnetic Fields interface, Input Sweep Settings in the Magnetic and Electric Fields interface, Manual Terminal Sweep Settings section in the Electrostatics interface and the Manual Terminal Sweep Settings section in the Electric Currents interface describe how to cycle through the ports and terminals, compute the entire S-matrix and export it to a Touchstone file. It is also possible to choose to export impedance (Z) or admittance (Y) matrices instead.

Connecting to Electrical Circuits

In this section:

- About Connecting Electrical Circuits to Physics Interfaces
- Connecting Electrical Circuits Using Predefined Couplings
- Connecting Electrical Circuits by User-Defined Couplings
- Solving
- Postprocessing

> *Inductor in an Amplifier Circuit*: Application Library path
> **ACDC_Module/Electromagnetics_and_Circuits/inductor_in_circuit**

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

In the AC/DC Module, the physics features supporting the connection to an electrical circuit are the **Terminal** and **Floating Potential** features in electric field interfaces and the **Coil** and **Lumped Port** features in magnetic field interfaces.

These features either accept a voltage measurement from the connecting noncircuit physics interface and return a current from an Electrical Circuit interface or the other way around.

> 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, interfaces in the AC/DC Module, RF Module, MEMS Module, Electric Discharge Module, Plasma Module, and Semiconductor 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 noncircuit 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 Terminals and Coils.

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 noncircuit physics interface.

- Define your own voltage or current measurement in the noncircuit 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, 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, **Current through device R1 (comp1.currents)**. The variable name is shown in the **Settings** window for **State**.
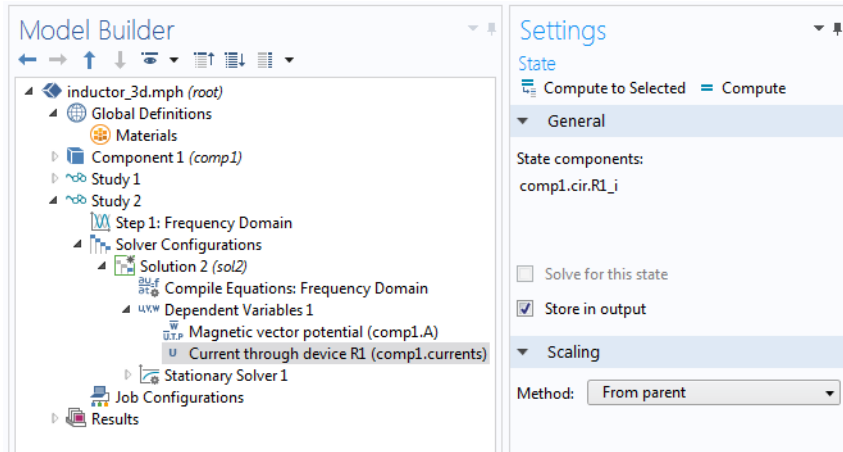
	Typically, voltage variables are named `cir.Xn_v` and current variables `cir.Xn_i`, where *n* is the "External" device number — 1, 2, and so on.

Solving

> ⚠ Some modeling errors lead to the error message *The DAE is structurally inconsistent* being displayed when solving. This error typically occurs from having an open current loop, from connecting voltage sources in parallel, or connecting current sources in series.
>
> In this respect, the predefined coupling features are also treated as (ideal) voltage or current sources. The remedy is to close current loops and to connect resistors in series with voltage sources or in parallel with current sources.

Postprocessing

The Electrical Circuits interface, unlike most of the other physics interfaces, solves for a relatively large number of global dependent variables (such as voltages and currents), instead of solving for a few space-varying fields (such as temperature or displacement). For this reason, the Electrical Circuit interface does not provide default plots when computing a study.

The physics interface defines a number of variables that can be used in postprocessing. All variables defined by the Electrical Circuit interface are of a global scope, and can be evaluated in a **Global Evaluation** node (under **Derived Values**). In addition, the time evolution or dependency on a parameter can be plotted in a **Global** plot (under a **ID Plot Group** node).

The physics interface defines a Node voltage variable for each electrical node in the circuit, with name cir.v_name, where cir is the physics interface Label and <name> is the node Name. For each two-pin component, the physics interface also defines variables containing the voltage across it and the current flowing through it. For resistors it adds a heat source variable as well.

> 🔍 In the *COMSOL Multiphysics Reference Manual:*
> • Derived Values, Evaluation Groups, and Tables and Global Evaluation
> • Plot Groups and Plots and Global

Part Libraries

In electromagnetics simulations, the studied devices might have complex geometry, such as motors. In the AC/DC Module, such complex geometry can be conveniently built using the **Part Library**.

The **Part Library** contains the following categories:

- Homogenized Multiturn Coils
- Magnetic Cores
- Rotating Machinery 2D
- Single Conductor Coils

The parts are built from partially parameterized sequences of geometry instructions. For example, you can load the `surface_mounted_magnet_internal_rotor_2d` part into a model and then specify the number of magnetic poles in rotor, the diameter of the shaft, and so on.

▥	For an example, see *Permanent Magnet Motor in 2D*: Application Library path **ACDC_Module/Devices,_Motors_and_Generators/ pm_motor_2d_introduction**

⊙	Part Libraries in the *COMSOL Multiphysics Reference Manual*.

Material Libraries

The AC/DC materials library contains a number of materials that assist in modeling electromagnetic components such as actuators, motors, loudspeakers, generators, and so on. Two branches of material, **AC/DC** ( ) and **Nonlinear Magnetic** ( ) in the **Material Library** ( ), are frequently used. The **AC/DC** branch includes typical materials for electromagnetics modeling and the **Nonlinear Magnetic** branch includes nonlinear magnetization curves for a large set of ferromagnetic alloys.

	For detailed information about Materials including the AC/DC Material Library and Nonlinear Magnetic Material Library, see the *COMSOL Multiphysics Reference Manual*.

The **Liquid Metals** folder in the AC/DC Material Library branch contains values for the Thermal conductivity, Electric conductivity, Dynamic viscosity, and Density for liquid Titanium, Steel, Iron, Nickel, Copper, Aluminum, Magnesium, Tin, Lithium, Sodium, Wood's metal, Potassium, Gallium, NaK (Sodium-potassium eutectic), and Mercury based on the book *Introduction to Magnetohydrodynamics* by P.A. Davidson, published by Cambridge University Press. Copyright (c) P.A. Davidson 2017. Reproduced with permission of Cambridge University Press through PLSclear.

The **Biological Tissues** folder in the AC/DC Material Library branch contains values for the Electric conductivity, the Relative permittivity, and the Reference temperature, together with the Relaxation times and Relative permittivity contributions of several Debye poles, for 54 biological tissue types. This data can be used together with the Multipole Debye dispersion material models, and is taken from Ref. 1.

The AC/DC Material Library also contains the material property data from the following company's products:

BOMATEC

NdFeB standard grades[2] (–, *M, H, SH, UH, EH, AH, BH, /S, M/S, H/S, SH/S, UH/S, EH/S, AH/S, BH/S, H/ST, SH/ST, UH/ST, EH/ST, AH/ST, BH/ST*), NdFeB (bonded, injection molded, and extruded), Ferrite (isotropic, anisotropic, and injection molded), SmCo (injection molded), SmFeN (injection molded), SmCo5 grades, Sm2Co17 grades, and AlNiCo (cast and sintered).

---

2. Here, the hyphen "–" refers to "normal" grades not having a particular suffix.

The foregoing materials are the property of BOMATEC AG. For product information concerning such materials, see www.bomatec.com/en/products. For other questions concerning such materials, please contact contact@bomatec.ch or www.bomatec.com/en/get-in-touch.

MAGNETIC SHIELD CORPORATION

MuMETAL® is the property of Magnetic Shield Corporation. For product information concerning this material, see https://www.magnetic-shield.com/. For other questions concerning this material, please contact https://www.magnetic-shield.com/contact-us/.

Reference

1. S. Salah ud din, *Acquisition of Dielectric Properties of Tissues in the Microwave Range*, doctoral dissertation, College of Engineering & Informatics, National University of Ireland Galway, 2019.

4

Electric Field and Current Interfaces

This chapter summarizes the functionality of the electric field and current interfaces found under the **Electric Fields and Currents** folder under the **AC/DC** branch ( ). 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, Boundary Elements Interface
- The Electric Currents Interface
- The Electric Currents in Shells Interface
- The Electric Currents in Layered Shells Interface
- The Electric Currents, Single Layer Shell Interface
- The Electric Currents, Layered Shell Interface

The Electrostatics Interface

The **Electrostatics (es)** interface (⬦), found under the **AC/DC** > **Electric Fields and Currents** branch when adding a physics interface, is used to compute the electric field, electric displacement field, and potential distributions in dielectrics under conditions where the electric charge distribution is explicitly prescribed. The formulation is stationary except for when it is used together with other physics interfaces. Eigenfrequency, frequency-domain, small-signal analysis, and time-domain modeling are supported in all space dimensions.

The physics interface solves Gauss' law for the electric field using the scalar electric potential as the dependent variable.

In addition to this standard formulation, the **Electrostatics** interface also has the option of using both the displacement field and the scalar potential as the dependent variables, see The D-V Formulation. This is set in the discretization settings for the electrostatics interface. The following sections in the **Electrostatics** interface chapter refer to the interface using the default **V-formulation** implementation. The information specific to the **D-V formulation** is given in the section The Electrostatics Interface Using the D-V Formulation.

Charge Conservation is the main node, 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 physics interface is added, these default nodes are also added to the **Model Builder** — **Free Space**, **Zero Charge** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and space charges. You can also right-click **Electrostatics** to select physics features from the context menu.

Physics-Controlled Mesh

The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the **Sequence type** is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the checkbox in the **Use** column on the same table row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

Information from the physics, such as the presence of an infinite elements domain or periodic condition, will be used to automatically set up an appropriate meshing sequence.

!	In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is es.

CROSS-SECTION AREA (1D COMPONENTS)

For 1D components, 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 components. See also Change Cross Section.

THICKNESS (2D COMPONENTS)

For 2D components, 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 components. See also Change Thickness (Out-of-Plane).

MANUAL TERMINAL SWEEP SETTINGS

Enter a **Reference impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω.

Select the **Use manual terminal sweep** checkbox 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.

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 a **Parameter format (value pairs)** for the Touchstone export — **Magnitude and angle (MA)** (the default), **Magnitude in dB and angle (DB)**, or **Real and imaginary parts (RI)**. Select an option from the **If file exists** list — **Overwrite** or **Create new**. Create new is useful when the model is solved multiple times with different settings. Select a **Parameter to export**: **Z** (the default), **Y**, or **S**.

From the **Home** toolbar add a **Parameters** node and enter the chosen name and assign to it a temporary **Expression** of unity into the **Parameters** table. You can also right-click **Global Definitions** to add the node.

| ! | 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. |

| 📝 | For most applications it is simpler to sweep over terminals using the Stationary Source Sweep study step. See Stationary Source Sweep in the *COMSOL Multiphysics Reference Manual* and (for more details) Stationary Source Sweep in the *AC/DC Module User's Guide*. |

DEPENDENT VARIABLES

The dependent variable is the **Electric potential** V. You can change its name, which changes both the field name and the variable name. If the new name coincides with the name of another dependent variable in the model, the physics interfaces shares degrees of freedom. The new name must not coincide with the name of a field of another type or with a component name belonging to some other field.

In addition to the standard **V formulation**, the interface also has the option to use the **D-V formulation**, which is mixed formulation method based on using both the electric displacement field **D** and the electric potential V as dependent variables. This option is set in the discretization setting of the **Electrostatics** interface.

DISCRETIZATION

Select the shape order for the **Electric potential** dependent variable — **Linear**, **Quadratic** (the default), **Cubic**, **Quartic**, or **Quintic**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

The **Mixed finite element** options set the **Electrostatics** interface to use the **D-V formulation**. This uses divergence elements to compute the displacement field which can be set to use either type 1 (not labeled) or type 2. Both of these types of divergence elements are available in linear and quadratic orders. For more information on the divergence elements, see Shape Function Types (Element Types).

!	The **Mixed Finite Element type 2** options only support simplex mesh elements.

!	When using the **Electrostatics** interface with the **Electromechanics** Multiphysics coupling, then the default discretization is **Mixed Finite Element (quadratic shape function)**.

!	In the *COMSOL Multiphysics Reference Manual*, see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

🔍	• Domain, Boundary, Edge, Point, and Pair Nodes for the Electrostatics Interface • Theory for the Electrostatics Interface

▦	*Electric Sensor*: Application Library path **COMSOL_Multiphysics/ Electromagnetics/electric_sensor**

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

The Electrostatics interface has these domain, boundary, edge, point, and pair nodes available.

ABOUT THE BOUNDARY CONDITIONS

The relevant physics 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 NODES

These nodes, listed in alphabetical order, are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users). Also see Table 4-1 for a list of interior and exterior boundary conditions, including edge, point, and pair availability.

> In general, to add a node, go to the **Physics** toolbar no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Archie's Law
- Change Cross Section
- Change Thickness (Out-of-Plane)
- Charge Conservation
- Charge Conservation, Piezoelectric[1,2]
- Charge Conservation, Ferroelectric[2,3]
- Conduction Loss (Time-Harmonic)
- Dispersion
- Dielectric Shielding
- Distributed Capacitance
- Effective Medium
- 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
- Symmetry Plane (for Electric Field)
- Terminal
- Thin Low Permittivity Gap
- Zero Charge (the default boundary condition)

[1]This feature is available with the Piezoelectricity multiphysics interface.

[2]Requires either the Acoustics Module, MEMS Module, or Structural Mechanics Module.

[3]This feature is available with the Ferroelectroelasticity multiphysics interface.

Table 4-1 lists the interior and exterior boundary conditions available with this physics 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.

NODE	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	pairs
Electric Displacement Field	x	x	pairs

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

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

For axisymmetric components, 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. There are also **Line Charge (on Axis)** and **Point Charge (on Axis)** available.

In the *COMSOL Multiphysics Reference Manual*, see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Free Space

For electromagnetic modeling, it is quite common for the fields to extend into the environment of the device in question. It is therefore important not only to model the device itself, but also its surroundings. Typically, this is done by adding a box or sphere around the device's geometry.

The default **Free Space** node is used to specify the physical conditions in close proximity to the device — typically in air or vacuum. It adds a **Charge Conservation** governing equation using the **Relative permittivity** constitutive relation with the permittivity ε_r set to one.

The **Free Space** feature provides a canvas on top of which other features can be added to locally specify material properties and excitation forms.

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. There are two types of **Charge Conservation** available; **Charge Conservation in Solids** and **Charge Conservation in Fluids**. This distinction decides how materials behave and how material properties are interpreted when the mesh is deformed.

Charge Conservation in Solids applies to materials whose properties change as functions of material strain, material orientation, and other variables evaluated in a material reference configuration (material frame).

Charge Conservation in Fluids applies to materials whose properties are defined only as functions of the current local state at each point in the spatial frame, and for which no unique material reference configuration can be defined.

!	In COMSOL versions **6.2** and earlier, this **Solids/Fluids** distinction was controlled within the **Charge Conservation** node by specifying **Solid/ Nonsolid** in the **Material type** setting.

CONSTITUTIVE RELATION D-E

Select a **Dielectric model** to describe the macroscopic properties of the medium (relating the electric displacement **D** with the electric field **E**) and the applicable material properties, such as the relative permittivity. Note that **Charge Conservation in Fluids** uses only **Relative permittivity** or **Polarization**. The other constitutive relations are only accessible in **Charge Conservation in Solids**. Select:

- **Relative permittivity** (the default) to use the constitutive relation $\mathbf{D} = \varepsilon_0\varepsilon_r\mathbf{E}$. Then the default is to take the **Relative permittivity** ε_r (dimensionless) values **From material**. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix. The default is 1. If **Effective medium** is selected, the Effective Medium subnode is available from the context menu (right-click the parent node) as well as from the **Physics** toolbar, **Attributes** menu.

- **Polarization** to use the constitutive relation $\mathbf{D} = \varepsilon_0\mathbf{E} + \mathbf{P}$. Then enter the components based on space dimension for the **Polarization** vector $\mathbf{P}$ (SI unit: C/m^2). The defaults are $0 \ C/m^2$.

- **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). Then the default is to take the **Relative permittivity** ε_r (dimensionless) values **From material**. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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). The defaults are $0 \ C/m^2$.

- **Dielectric losses** to use the constitutive relation $\mathbf{D} = \varepsilon_0(\varepsilon' - j\varepsilon'')\mathbf{E}$. Specify that the **Relative permittivity (real part)** ε' (dimensionless) and the **Relative permittivity (imaginary part)** ε'' (dimensionless) must be taken **From material** or be **User defined**. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix. The default is 1. The material parameters **Relative permittivity (real part)** ε' and the **Relative permittivity (imaginary part)** ε'' form the complex relative permittivity $\varepsilon_r = \varepsilon' - j\varepsilon''$.

> ⚠️ Note that the time-harmonic Sign Convention requires a lossy material to have a positive material parameter ε'' (see Modeling Losses in the Frequency Domain).

- **Ferroelectric** to use the constitutive relation $\mathbf{D} = \varepsilon_0\mathbf{E} + \mathbf{D_r} + \mathbf{P(E)}$, where $\mathbf{D_r}$ is the remanent electric displacement, and the polarization vector is $\mathbf{P}$ calculated from the electric field using a nonlinear relation of ferroelectricity with possible saturation and hysteresis. This option is available when the **Material type** is set to **Solid**.

> 🔍 Ferroelectricity

- **Dispersion** to use the constitutive relation $\mathbf{D} = \varepsilon_0\mathbf{E} + \mathbf{P(E}, \varepsilon_{rS})$, where the polarization vector is calculated from the electric field using the dielectric dispersion model. This option is available when the **Material type** is set to **Solid**. You enter the **Relative permittivity** ε_{rS} (dimensionless) values **From material**. For **User defined**, select **Isotropic**, **Diagonal**, or **Symmetric** and enter values or expressions in the field or

matrix. This value of electric permittivity will be used in stationary study, for which the polarization vector is calculated as $\mathbf{P} = \varepsilon_0(\varepsilon_{rS} - I)\mathbf{E}$.

🔍 Once this option is selected, a subnode Dispersion will become available under the **Charge Conservation** node. At that subnode, you can select the dispersion model, enter the corresponding parameters, and choose how the relative permittivity input on the parent node will be interpreted in Eigenfrequency, Frequency Domain, and Time Dependent studies.

Conduction Loss (Time-Harmonic)

📋 This feature requires either the Acoustics Module, or MEMS Module, or Structural Mechanics Module. See the individual documentation for information.

The **Conduction Loss (Time-Harmonic)** subnode to **Charge Conservation** allows you to model possible conductive losses in a dielectric material. The effect is only active in an Eigenfrequency or Frequency Domain study.

CONDUCTION CURRENT

By default, the **Electric conductivity** σ for the media is defined **From material**. You can also select **User defined** or **Linearized resistivity**.

- For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** depending on the characteristics of the electric conductivity, and then enter values or expressions for the **Electric conductivity** σ in the field or matrix.

- For **Linearized resistivity** the default **Reference temperature** T_{ref}, and **Resistivity temperature coefficient** α, and **Reference resistivity** ρ_0 are taken **From material**, which means that the values are taken from the domain (or boundary) material. 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.

LOCATION IN USER INTERFACE

Context Menus
Electrostatics > **Charge Conservation** > **Conduction Loss (Time-Harmonic)**

Physics tab with **Charge Conservation** node selected in the model tree:

Attributes > **Conduction Loss (Time-Harmonic)**

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.

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.

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 physics interface defines.

SPACE CHARGE DENSITY

Enter a value or expression for the **Space charge density** ρ_v (SI unit: C/m^3). The default is 0 C/m^3.

Force Calculation

Use the **Force Calculation** node to define globally available force and torque variables for the selected domains. **Force Calculation** is also available and defined for the **Magnetic Fields**, **Magnetic and Electric Fields**, **Magnetic Field Formulation**, and **Rotating Machinery, Magnetic** interfaces.

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 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. For the **Magnetic Fields**, **Magnetic and Electric Fields**, and **Rotating Machinery, Magnetic** interfaces, the force calculation includes both electric and magnetic forces. The computed force is made available as a global vector variable with name `<name>.Force<component>_<force name>`, where `<name>` is the **Name** of the physics interface.

Enter a direction vector for the **Torque axis r_{ax}** and coordinates for the **Torque rotation point r_0**. A torque calculation about a given point (**Torque rotation point**) is made and defined as a global vector variable, `<name>.T<component>_<force name>`. The resulting torque component parallel to the given **Torque axis** is given as a global scalar variable, typically `<name>.Tax_<force name>`.

In frequency domain studies, the cycle-averaged force and torque variables are also defined, with names `<name>.Forceav<component>_<force name>`, `<name>.Tav<component>_<force name>` and `<name>.Taxav_<force name>`. The variables described are still defined and represent the instantaneous force and torque.

⚠️   • In 2D axisymmetry, the radial component of the total force is set to zero, as this is an intrinsic property of the used symmetry.

    • In small-signal analysis studies, no cycle-average variables are defined. Use the additional plot settings for the Harmonic Perturbation on the instantaneous variables to compute the quantities of interest. See Harmonic Perturbation, Prestressed Analysis, and Small-Signal Analysis in the *COMSOL Multiphysics Reference Manual*.

🔍   For more information on calculation of forces and torques, see Computing Forces and Torques.

ADVANCED SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Compute both instantaneous and cycle-averaged variables in the frequency domain** checkbox is selected by default. This checkbox controls which variables are defined by the Force Calculation feature in frequency-domain studies. When selected (the default setting in new models), the variables are defined as described above. This checkbox selected by default for backward compatibility. If the checkbox is cleared, only the cycle-averaged variables are computed, replicating the behavior of the Force Calculation feature in version 4.3 and earlier. The checkbox is cleared in existing Force Calculation features when opening models created in version 4.3 and earlier.

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.

Ground

The **Ground** node implements ground (zero potential) 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 physics interfaces, also select additional **Ground** nodes from the **Edges** (3D components) or **Points** (2D and 3D components) submenus. For 2D axisymmetric components, it can be applied on the Symmetry axis.

BOUNDARY, EDGE, OR POINT SELECTION

⚠️ 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.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

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 physics interface, the value of the potential is typically defined at some part of the geometry. For some physics interfaces, also select additional **Electric Potential** nodes from the **Edges** (3D

components) or **Points** (2D and 3D components) submenus. For 2D axisymmetric components, it can be applied on the symmetry axis.

⚠️ | 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.

ELECTRIC POTENTIAL

Enter the value or expression for the **Electric potential** V_0 (SI unit: V). The default is 0 V.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

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.

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or click **Harmonic Perturbation** in the **Physics** toolbar. For more information, see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

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.

MATERIAL TYPE

The **Material type** setting decides how materials behave and how material properties are interpreted when the mesh is deformed. Select **Solid** for materials whose properties change as functions of material strain, material orientation, and other variables evaluated in a material reference configuration (material frame). Select **Nonsolid** for materials whose properties are defined only as functions of the current local state at each point in the spatial frame, and for which no unique material reference configuration can be defined. Select **From material** to pick up the corresponding setting from the domain material on each domain.

EXTERNAL SURFACE CHARGE ACCUMULATION

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

Symmetry Plane (for Electric Field)

The **Symmetry Plane** node adds a boundary condition that represents symmetry or antisymmetry in the electric field, depending on which option is chosen.

SYMMETRY TYPE

Choose between **Symmetry** and **Antisymmetry** for the electric field. If **Symmetry** is chosen, the boundary condition is

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

which states that the normal component of the electric field is zero. If **Antisymmetry** is chosen, the boundary condition becomes

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

which states that the tangential component of the electric field is zero.

> Dielectric Shielding Comparison: Application Library path **ACDC_Module/ Introductory_Electrostatics/dielectric_shielding_comparison**

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.

ELECTRIC DISPLACEMENT FIELD

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

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. For more complex geometries, it might be necessary to use the **Destination Selection** section, which you activate by right-clicking the **Periodic Condition** node and select **Manual Destination Selection**. With that section, the boundaries which constitute the source and destination surfaces can be manually specified.

> When this feature is used in conjunction with a Sector Symmetry feature on connected boundaries, wherever the sector symmetry boundaries connect with the periodic boundaries, the same periodic condition feature cannot be used on both sides. At least two periodic condition features are required for the model to compute correctly.

The software usually automatically identifies the boundaries as either source boundaries or destination boundaries, as indicated in the selection list. This works fine for cases like opposing parallel boundaries. In other cases, right-click **Periodic Condition** and select **Manual Destination Selection** to control the destination. By default it contains the selection that COMSOL Multiphysics identifies.

When using nonconforming meshes on the source and destination of a periodic boundary pair, for numerical stability, a finer mesh should be applied on the destination side. Use conforming meshes if possible.

DESTINATION SELECTION

This section is available for specifying the destination boundaries, if needed, when the **Manual Destination Selection** option is selected in the context menu for the **Periodic Condition** node. You can only select destination boundaries from the union of all source and destination boundaries.

PERIODIC CONDITION

Select a **Type of periodicity** — **Continuity** (the default), **Antiperiodicity**, or **Floquet periodicity**. Select:

- **Continuity** to make the potential periodic (equal on the source and destination).
- **Antiperiodicity** to make it antiperiodic.
- **Floquet periodicity** (only available with products supporting piezoelectric modeling). Specify the components of the **k-vector for Floquet periodicity** $\mathbf{k}_F$ (SI unit: rad/m).

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

ORIENTATION OF SOURCE

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

This section appears if the setting for **Transform to intermediate map** in the **Orientation of Source** section is changed from the default value, **Automatic**, and **Advanced Physics Options** is selected in the **Show More Options** dialog. For information about the **Orientation of Destination** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

Thin Low Permittivity Gap

Use the **Thin Low Permittivity Gap** node

$$\mathbf{n} \cdot \mathbf{D}_1 = \frac{\varepsilon_0 \varepsilon_{\mathbf{r}}}{d}(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{D}_2 = \frac{\varepsilon_0 \varepsilon_{\mathbf{r}}}{d}(V_2 - V_1)$$

to model a thin gap of a material with a small permittivity compared to the adjacent domains. The layer has the thickness d and the relative permittivity $\varepsilon_{\mathbf{r}}$. The indices 1 and 2 refer to the two sides of the boundary.

THIN LOW PERMITTIVITY GAP

The default is to take the **Relative permittivity** $\varepsilon_{\mathbf{r}}$ (dimensionless) values **From material**. For **User defined**, enter a different value or expression. Enter a **Thickness** d (SI unit: m). The default is 5 mm.

▦	*Thin Low Permittivity Gap Comparison*: Application Library path **ACDC_Module/Introductory_Electrostatics/ thin_low_permittivity_gap_comparison**

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 $\varepsilon_{\mathbf{r}}$ that shields the electric field:

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

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

MATERIAL TYPE

The **Material type** setting decides how materials behave and how material properties are interpreted when the mesh is deformed. Select **Solid** for materials whose properties change as functions of material strain, material orientation and other variables evaluated in a material reference configuration (material frame). Select **Nonsolid** for materials whose properties are defined only as functions of the current local state at each point in the spatial frame, and for which no unique material reference configuration can be defined. Select **From material** to pick up the corresponding setting from the domain material on each domain.

ELECTRIC SHIELDING

The default is to take the **Relative permittivity** ε_r (dimensionless) values **From material**. It takes it from the adjacent domains if not explicitly defined. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix. Enter a **Surface thickness** d_s (SI unit: m) of the shielding. The default is 1 m.

	For a dielectric shielding example, see *Dielectric Shielding Comparison*: Application Library path **ACDC_Module/Introductory_Electrostatics/ dielectric_shielding_comparison**.

Terminal

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

⚠	In most cases, it is not recommended to include the interior of good conductors in the simulation. When using the **Terminal** on the domain level, the interior unknowns for the electric potential are automatically replaced by a single constant, thus avoiding numerical problems arising from a very high material contrast. If one instead uses the **Terminal** on the boundary level, it is recommended to exclude the interior of the associated conductor from the domain selection for the main Electrostatics node.

An operation called **Split by Connectivity** (✎) is available by right-clicking the parent node. The operation is useful when there are several terminal domains or boundaries

that are not geometrically connected. Clicking the operation node will automatically generate several **Terminal** nodes where each selection is geometrically connected.

The **Terminal** node also enables the computation of the lumped parameters of the system, such as capacitance. See Computing Lumped Parameters.

TERMINAL

Specify the terminal properties. Enter a unique **Terminal name** that will be used to identify the global variables (such as the terminal voltage or current) that are defined by the terminal. The **Terminal name** can contain alphanumeric characters, but it should be numeric when performing terminal sweeps.

Select a **Terminal type** — **Charge** (the default), **Voltage**, **Circuit**, **Terminated (power)**, or **Terminated (voltage)**. Select:

- **Charge** to prescribe the total charge deposited on the terminal boundaries. Enter the total charge Q_0. The default is 0 C, which corresponds to a uncharged floating electrode.

- **Voltage** to specify a fixed electric potential at the terminal. Enter an electric potential V_0. The default is 1 V.

⚠️   To correctly calculate the terminal charge and the corresponding capacitances, if **Voltage** Terminal is used on boundaries between The Electrostatics Interface and The Electrostatics, Boundary Elements Interface, let the dependent variable for The Electrostatics, Boundary Elements Interface be the same as the dependent variable for The Electrostatics Interface and disable Electric Scalar-Scalar Potential Coupling. How to set the name for the dependent variable is described in the Dependent Variables section. Since the Electric Scalar-Scalar Potential Coupling is disabled, add manually a Fully Coupled node to the Stationary Solver below the Solution node in the Solver Configurations.

- **Circuit** to specify a terminal connected to an Electrical Circuit interface that models an external circuit. The Terminal node provides a current-voltage characteristic to the circuit.

> **!** The Electrical Circuit Interface requires a current from the Terminal, so the **Circuit** terminal type can only be used in Time Dependent or Frequency Domain studies, in which the current can be computed as the time derivative of the charge.
>
> The **Circuit** type cannot be used together with a terminal sweep.

- **Terminated (power)** to connect the terminal to an impedance that might represent a load or a transmission line. When **Terminated (power)** is selected, the scattering parameters (S-parameters) are computed. The **Terminated (power)** setting can only be used in frequency-domain studies.

  - Enter a **Terminal power** P_0 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**. For **Physics interface**, the **Reference impedance** Z_{ref} defined in the **Settings** window for the physics under Sweep Settings is used. For **User defined**, enter a specific impedance Z_{ref} for this terminal. The default is 50 Ω.

- **Terminated (voltage)** to connect the terminal to an impedance that might represent a load or a transmission line. The **Terminated (voltage)** option is essentially the same as **Terminated (power)**, with the power defined as: $P_0 = 0{,}5 \cdot |V_0|^2 / Z_{\mathrm{ref}}$.

When **Terminal type** is selected as **Charge** or **Voltage**, a contribution can be added as **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

Initial values
The initial values subsection is used to provide initial values for the internal state variables in the Terminal. These settings can be useful to provide an initial guess to the nonlinear solver or the initial value in a time-dependent study.

The **Initial value for voltage** is available for all Terminal types except Voltage. Enter an **Initial value for voltage** V_{init}.

The **Initial value for charge** is available for the Circuit excitation. It specifies the initial charge on the terminal when the circuit is connected. Enter an **Initial value for charge** Q_{init}.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

🔍	Studies and Solvers in the *COMSOL Multiphysics Reference Manual*

▥	For a voltage terminal type example, see *Dielectric Shielding Comparison*: Application Library path **ACDC_Module/ Introductory_Electrostatics/dielectric_shielding_comparison**.

Floating Potential

The **Floating Potential** node is used when modeling a metallic electrode at floating potential. It applies a constant voltage V_0 on the boundary (for domain features, this is the boundary enclosing the selected domain), such that the total normal electric displacement field **D** equals a specific charge Q_0:

$$V = V_0$$

$$\int_{\partial\Omega} (\mathbf{D} \cdot \mathbf{n})dS = Q_0,$$

where $\partial\Omega$ represents the boundary and **n** refers to the surface normal. The constant boundary voltage implies the tangential electric field equals zero; the electric field will be perpendicular to the boundary:

$$-\nabla_t V = \mathbf{E}_t = 0$$

$$\mathbf{n} \times \mathbf{E} = 0.$$

In case of $Q_0 = 0$ (the default case), the boundary will behave as an unconnected, neutrally charged, good conductor under electrostatic conditions (a floating equipotential). Although locally the displacement field may vary, the total electric displacement field entering or leaving the boundary equals zero.

If the floating potential touches a point, boundary, or domain feature that is not floating (a **Terminal** or **Ground** feature), the floating potential will acquire that feature's potential. If the floating potential is set to a certain charge, or connected to a circuit, it behaves like a terminal.

FLOATING POTENTIAL

The **Floating potential identifier** text area shows the unique identifier for the floating potential feature. It is used to identify the global variables created by the node, such as the voltage. The **Floating potential group** checkbox in the **Settings** window for **Floating Potential** controls how potentials are assigned to boundary segments in the feature's boundary selection. If this checkbox is not selected, a single potential is applied to all boundaries in the selection. If the checkbox is selected, each group of contiguous boundaries in the selection is given a unique potential. This simplifies the setup of models with many floating electrodes. The values of the potential at each group of boundaries are then made available in postprocessing, collected in a vector variable.

The following options are not available if the **Floating potential group** checkbox is selected. Select a specification for the **Electric charge** — choose **User defined** to specify a total **Charge** Q_0 deposited on the surface. Select **Circuit** to connect the floating potential to an Electrical Circuit.

> ! The Electrical Circuit Interface requires a current from the Floating Potential, so the **Circuit** terminal type can only be used in Time Dependent or Frequency Domain studies, in which the current can be computed as the time derivative of the charge.

Initial Values

The initial values subsection is used to provide initial values for the internal state variables in the Floating potential. These settings can be useful to provide an initial guess to the nonlinear solver or the initial value in a time-dependent study.

Enter an **Initial value for voltage** V_{init}.

The **Initial value for charge** is available for the Circuit excitation. It specifies the initial charge on the floating potential when the circuit is connected. Enter an **Initial value for charge** Q_{init}.

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

🔍	Settings and Properties Windows for Feature Nodes in the *COMSOL Multiphysics Reference Manual*

Distributed Capacitance

The **Distributed Capacitance** node adds a distributed capacitance boundary condition according to the following equations for exterior boundaries:

$$-\mathbf{n} \cdot \mathbf{D} = \varepsilon_0 \varepsilon_r \frac{V_{\text{ref}} - V}{d_s}$$

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}.

DISTRIBUTED CAPACITANCE

Enter the values or expressions for **Relative permittivity** ε_r (dimensionless), **Surface thickness** d_s (SI unit: m), and **Reference potential** V_{ref}. The default value for the surface thickness is 10^{-3} m (1 mm) and 0 V for the reference potential.

Line Charge

For 3D components, use the **Line Charge** node to specify line charges along the edges of a geometry. Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

EDGE SELECTION

⚠	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 and the default is 0 C/m.

| ⊕ | Line Charge (on Axis) and Line Charge (Out-of-Plane) |

Line Charge (on Axis)

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

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or click **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

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 and the default is 0 C/m.

| ⊕ | Line Charge and Line Charge (Out-of-Plane) |

Line Charge (Out-of-Plane)

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

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

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

⚠️ 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 and the default is 0 C/m.

🔍 Line Charge and Line Charge (on Axis)

Point Charge

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

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

POINT SELECTION

⚠️ 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. The default is 0 C.

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

Point Charge (on Axis)

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

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

POINT SELECTION

> ⚠ 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 an axis. This source represents an electric displacement field flowing out of the point. The default is 0 C.

> 🔍 Point Charge and Line Charge (Out-of-Plane)

Change Cross Section

This node is available with 1D components. This setting overrides the global **Cross-Section Area** setting made in any physics interface that uses this feature. For 2D components, see Change Thickness (Out-of-Plane).

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

CHANGE CROSS SECTION

Enter a value or expression for the **Cross-section area** A. The default value of one unit area is typically not representative for the actual domain. Instead, it describes a unit area that makes the 1D equation identical to the equation used for 3D components.

Change Thickness (Out-of-Plane)

This node is available for 2D components. This setting overrides the global **Thickness** setting made in any physics interface that uses this node. For 1D components, see Change Cross Section.

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

CHANGE THICKNESS (OUT-OF-PLANE)

Enter a value or expression for the **Out-of-plane thickness** d (SI unit: m). The default value is, in most cases, 1 unit length, which 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 components.

Charge Conservation, Ferroelectric

This feature is available with the Ferroelectroelasticity multiphysics interface which requires either the Acoustics Module, MEMS Module, or Structural Mechanics Module. See Ferroelectricity and Ferroelectroelasticity (in the *Structural Mechanics Module User's Guide*) for more information.

The **Charge Conservation, Ferroelectric** node is normally used together with a **Electrostriction** multiphysics coupling node and **Solid Mechanics** interface. The node is added by default to the **Electrostatics** interface when adding a Ferroelectroelasticity interface. It is also available from the context menu (right-click the **Electrostatics** interface parent node) or from the **Physics** toolbar.

Charge Conservation, Piezoelectric

This feature is available with the Piezoelectricity interface, which requires either the Acoustics Module, or MEMS Module, or Structural Mechanics Module. See the individual documentation for information.

The **Charge Conservation, Piezoelectric** node is normally used together with a **Piezoelectric Effect** multiphysics coupling node and a corresponding **Piezoelectric**

Material node in the **Solid Mechanics** interface. The node is added by default to the **Electrostatics** interface when adding a Piezoelectricity interface. It is also available from the context menu (right-click the **Electrostatics** interface parent node) or from the **Physics** toolbar.

> ⚠️ When the **Charge Conservation, Piezoelectric** node is added to the electrostatics interface in the absence of an active **Piezoelectric Effect** multiphysics coupling node, the material behaves similarly to a Charge Conservation node, with electric properties corresponding to the relative permittivity entered (see below). The piezoelectric effect is *not* included in the corresponding equation system.

ELECTRIC DISPLACEMENT

If the node is used together with an active **Piezoelectric Effect** multiphysics coupling node, then these settings are locked. Note that if they are unlocked, then the material behaves like a dielectric and *not* a piezoelectric. In this case, the default is to take the **Relative permittivity** ε_{rS} (dimensionless) values **From material**. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix.

Dispersion

The **Dispersion** subnode to **Charge Conservation** and **Charge Conservation, Piezoelectric** allows you to model possible dielectric losses in the material. The effect can be accounted for in Eigenfrequency, Frequency Domain and Time Dependent study.

Under the **Material model**, you can select one of the following option:

- **Debye** (default)
- **Multipole Debye**
- **Cole–Cole**
- **Havriliak–Negami**
- **Constant Loss Tangent**

- **User defined**

> The **Dispersion** subnode is contributing in a sense that the polarization contributions from two or more such subnodes added under the same parent node are additive.
>
> Thus, you can add several **Dispersion** subnodes and even set them to use different dispersion models to build up an effective multipole model of any degree of complexity. For example, the Multipole Cole-Cole model can be set up this way.
>
> In COMSOL versions **6.2** and earlier, the **Dispersion** node was exclusive in a sense so that only one subnode under the same parent was effective on the selected domain.

DEBYE AND MULTIPOLE DEBYE MODEL

For a single pole **Debye** dispersion model, you can enter the **Relaxation time** and **Relative permittivity contribution**. These parameters can be either scalar, or diagonal matrices to account for possible anisotropy in the material.

For the **Multipole Debye** model, you can enter any number of **Relaxation time** and the corresponding **Relative permittivity contribution** values using an input table with dynamically changed size. However, only scalar values can be entered.

Both models are directly suitable for Eigenfrequency, Frequency Domain and Time Dependent study types. And for these study types, you can specify how the relative permittivity input, ε_{rS}, on the parent node should be interpreted by selecting the **Static response** (the default value is **Low frequency limit**).

In the Thermal Effects section, you can select the **Shift function** type, which will be used to compute the effective relaxation time accounting for the temperature effects.

CONSTANT LOSS TANGENT

In this case, you enter the **Loss Tangent** $\eta(f_c)$ together with the **Center frequency** f_c. You also specify the model **Bandwidth (decades)** that defines a frequency interval centered at f_c, in which the loss tangent will be approximately constant and equal to $\eta(f_c)$. You can also select the **Accuracy** for the approximation to be either **Normal** (default) or **High**. The software will use an equivalent Multipole Debye model and automatically deduce the necessary number of poles together with the values of the corresponding relaxation

times and relative permittivity contributions, which will be used in computations to maintain the requested bandwidth and accuracy.

🔍	Dispersion Theory

⚠	In Eigenfrequency study and Time Dependent study, each pole will require one extra vector degree of freedom (domain state variable) to represent the corresponding contribution to the polarization. Thus, using many poles in the relaxation data input table for **Multipole Debye** model, or setting the **Accuracy** to **High** in case of **Constant Loss Tangent**, can lead to significant computation costs for larger models.

COLE-COLE AND HAVRILIAK–NEGAMI MODELS

These fractional models are extension of the Debye model. In some cases, they can provide a better representation of the relaxation spectrum.

For the **Cole-Cole** model, you enter a fractional order parameter, the **Complementary broadening parameter** α_{CC} that controls a symmetric (in log-space) broadening of the spectrum. Note that its acceptable value is limited to the range from 0 to 1.

For the **Havriliak-Negami** model, you can enter two fractional order parameters, the **Symmetric broadening parameter** β_{HN} with the value limited to the range from 0 to 1, and the **Asymmetric broadening parameter** γ_{HN} with the value limited to the range from 0 to $1/\beta_{HN}$.

At the **Time Domain and Eigenfrequency** section, you can compute an approximation for the complex valued relative permittivity contribution.

By default, the approximation is not computed. Without it, the **Cole-Cole** and **Havriliak-Negami** models are limited to Frequency Domain study type only.

The approximation computation needs to be done as a preprocessing step. Use the **Compute approximation** (📈) action button that is available at the **Time Domain and Eigenfrequency** section. Once the computation has been performed, you can preview it using the **Preview plot** (〰) action button that will become active at the section. The approximation corresponds to the Multipole Debye model. You can check the **Show approximation data** checkbox to inspect the computed parameter values.

Thus computed approximation will be used in Eigenfrequency and Time Dependent study steps. For the eigenfrequency computations, the eigenvalues problem will be always linear.

For all study types, you can specify how the relative permittivity input, ε_{rS}, on the parent node should be interpreted by selecting the **Static response** (the default value is **Low frequency limit**).

USER DEFINED MODEL

Enter expressions for the real and imaginary parts of the relative permittivity contribution, $\Delta\varepsilon'$ and $\Delta\varepsilon''$. Also enter the value of the real part of the contribution in the low frequency limit $\Delta\varepsilon'(0)$. Note that the imaginary part of the contribution is always assumed to be zero in the low frequency limit.

The expressions can be entered as functions taken directly from interpolated data, or can be analytical expressions of the frequency variable.

!   The internal variables for the frequency f and angular frequency ω are named `phys.freq` and `phys.omega`, respectively. Here, `phys` is the tag of the parent physics (for instance, `solid`). You can also use the variable `freq` defined by the solver.

At the **Time Domain and Eigenfrequency** section, you can compute an approximation for the complex valued relative permittivity contribution.

By default, the approximation is not computed. Without it, the **User defined** model is limited to Frequency Domain study type only.

The approximation computation needs to be done as a preprocessing step. Use the **Compute approximation** (⬚) action button that is available at the **Time Domain and Eigenfrequency** section. Once the computation has been performed, you can preview it using the **Preview plot** (⬚) action button that will become active at the section. The approximation corresponds to the Multipole Debye model. You can check the **Show approximation data** checkbox to inspect the computed parameter values.

Thus computed approximation will be used in Eigenfrequency and Time Dependent study steps. For the eigenfrequency computations, the eigenvalues problem will be always linear.

Note that for the **User defined** dispersion model, the relative permittivity input, ε_{rS}, on the parent node will be interpreted should be interpreted as the low frequency limit.

THERMAL EFFECTS

Dispersion properties depend on the temperature. For many materials, a change in the temperature can be transformed directly into a change in the time scale. Thus, the relaxation time is modified to $a_T(T)\tau_m$, where $a_T(T)$ is a shift function.

Select a Shift function — **None**, **Vogel–Fulcher**, **Arrhenius**, **Williams–Landel–Ferry**, **Tool–Narayanaswamy–Moynihan**, or **User defined**.

- When the default, **None**, is kept, the shift function $a_T(T)$ is set to unity and the relaxation time is not modified.

- For **Vogel–Fulcher** enter values or expressions for these properties:
  - **Reference temperature** T_0. The default is 230 K.
  - **Activation energy** Q. The default is 8000 J/mol.

- For **Arrhenius** enter values or expressions for these properties:
  - **Reference temperature** T_0. The default is 293.15 K.
  - **Activation energy** Q.

- For **Williams–Landel–Ferry** enter values or expressions for these properties:
  - **Reference temperature** T_{WLF} The default is 293.15 K.
  - **WLF constant 1** $C_{1\text{WLF}}$. The default is 17.44.
  - **WLF constant 2** $C_{2\text{WLF}}$. The default is 51.6 K.

- For **Tool–Narayanaswamy–Moynihan** enter values or expressions for these properties:
  - **Reference temperature** T_0. The default is 293.15 K.
  - **Activation energy** Q.
  - **Activation energy fraction** χ $(0 < \chi < 1)$.
  - **Structural relaxation times** λ_0.
  - **Fictive temperature weights** w.

- For **User defined** enter a value or expression for the shift function a_T.

DISCRETIZATION

To display this section, click the **Show More Options** button (👁) and select **Discretization** from the **Show More Options** dialog. Select the element order from the list box for the auxiliary electric field vector variables $\mathbf{e}_n$.

You can change the solution algorithm by using the checkbox **Use local time integration** (checked by default). When unchecked, the solution of the equations for the auxiliary

dependent variables in Time Dependent study will be governed by the time stepper algorithm used in the solver.

Electrostatic Point Dipole

Add an **Electrostatic Point Dipole** node to 3D and 2D components. Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or click **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

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.

ELECTROSTATIC POINT DIPOLE

Select a **Dipole specification** — **Magnitude and direction** (the default) or **Electric dipole moment**.

For **Magnitude and direction** enter coordinates for the **Electric dipole moment direction** $\mathbf{n}_p$ (dimensionless) and the **Electric dipole moment, magnitude** p.

For **Electric dipole moment** enter coordinates for the **Electric dipole moment p**.

The Electrostatics Interface Using the D-V Formulation

The **D-V formulation** is an option in the **Electrostatics** interface and is available with either the **ACDC** or **MEMS** module licenses. The **Electrostatics Interface** can be set to use

the **D-V Formulation** in the discretization settings for the interface. For background on the **D-V Formulation**, see the Theory for the Electrostatics Interface with the D-V Formulation section.

Once the formulation is set in the discretization menu, the rest of the **Electrostatics** interface is designed to operate in the same manner as default the **V-formulation**. However, the **D-V formulation** does not support the all features included in the standard **Electrostatics** interface.

> [!] The **D-V Formulation** is only available for 2D and 3D components.

The features not currently supported by the **D-V formulation** are:

- Pair features,
- Surface Charge Accumulation,
- External Surface Charge Accumulation,
- Change Thickness (out-of-plane).

The out-of-plane thickness can be set as a global property of the electrostatics interface, however a spatially dependent out-of-plane thickness is not supported.

Archie's Law

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 the subnodes are made available from the context menu (right-click the parent node) as well as from the **Physics** toolbar, **Attributes** menu.

Use the **Archie's Law** subnode to provide an electric conductivity computed using Archie's Law. This subnode can be used to model nonconductive effective medium saturated (or variably saturated) by conductive liquids, using the relation:

$$\sigma = s_L^n \varepsilon_p^m \sigma_L$$

> [Q] Archie's Law Theory

By default, the **Electric 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.

For **User defined** 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 dimensionless parameters as needed:

- **Cementation exponent** m
- **Saturation exponent** n
- **Fluid saturation** S_L
- **Porosity** ε_p to set up the volume fraction of the fluid.

Effective Medium

This subfeature is available only when **Effective medium** is selected as the material parameter (for example, **Relative permeability** or **Relative permittivity**) in the parent feature node when it is available with the physics interface (for example, the **Charge Conservation**, **Current Conservation**, or **Ampere's Law** nodes). Then the subnodes are made available from the context menu (right-click the parent node) as well as from the **Physics** toolbar, **Attributes** menu.

Use the **Effective Medium** subfeature to specify the material properties of a domain consisting of a porous medium using a mixture model. Depending on the specific physics interface being used, the subfeature 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.

EFFECTIVE MEDIUM

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 be fractional (between 0 and 1) and should add to 1 in normal cases.

	The availability of the **Effective Electric Conductivity**, **Effective Relative Permittivity**, and **Effective Relative Permeability** sections depend on the material properties used in the physics interface. In addition, these sections are only active if **Effective medium** is selected in the corresponding material property for the parent feature node.

EFFECTIVE ELECTRIC 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 (for example, **conductivity** or **permittivity**), the volume average of its inverse (for example, the **resistivity**), or the **Power law**. For each material, specify either **From material**, to take the value from the corresponding material specified in the **Effective Medium** section, or **User defined** to manually input a value.

	Effective Material Properties in Effective Media and Mixtures

The Electrostatics, Boundary Elements Interface

The **Electrostatics, Boundary Elements (esbe)** interface (⊞), found under the **AC/DC > Electric Fields and Currents** branch when adding a physics interface, is used for computing the potential distribution in dielectrics under conditions where the electric potential distribution on the boundaries is explicitly prescribed. The formulation is based on the boundary element method and the interface is available in 2D and 3D.

The physics interface solves Laplace's equation for the electric potential using the scalar electric potential as the dependent variable

$$-\nabla \cdot (\varepsilon_0 \varepsilon_r \nabla V) = 0 \,.$$

📝	For a discussion about the boundary element method, see Theory for the Boundary Elements PDE in the *COMSOL Multiphysics Reference Manual*.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Charge Conservation**, **Zero Charge** (on exterior boundaries and in 3D also on interior edges) and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, electric potential and surface charge density conditions. You can also right-click **Electrostatics, Boundary Elements** to select physics features from the context menu.

📝	If both The Electrostatics Interface and The Electrostatics, Boundary Elements Interface are available, the Electric Scalar-Scalar Potential Coupling node is available from the **Multiphysics** menu in the **Physics** toolbar or by right-clicking the **Multiphysics Couplings** node in **Model Builder**.
📝	The Electrostatics Interface and The Electrostatics, Boundary Elements Interface can also be coupled by using the same name for the dependent variable for both interfaces. Then Electric Scalar-Scalar Potential Coupling is not needed. How to set the name for the dependent variable is described in the Dependent Variables section.

Physics-Controlled Mesh

The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the **Sequence type** is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the checkbox in the **Use** column on the same table row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

Information from the physics will be used to automatically set up an appropriate mesh sequence.

> **!** In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh.

DOMAIN SELECTION

From the **Selection** list, select any of the options — **Manual**, **All domains**, **All voids**, or **All domains and voids** (the default). The geometric entity list displays the selected domain entity numbers. Edit the list of selected domain entity numbers using the selection toolbar buttons to the right of the list or by selecting the geometric entities in the **Graphics** window. Entity numbers for voids can be entered by clicking the Paste (⬚) button in the selection toolbar and supplying the entity numbers in the dialog. The entity number for the infinite void is 0, and finite voids have negative entity numbers.

Selections can also be entered using the **Selection List** window, available from the **Windows** menu in the **Home** toolbar.

> **▤** For more information about making selections, see Working with Geometric Entities in the *COMSOL Multiphysics Reference Manual*.

PHYSICS SYMBOLS

Select the **Enable physics symbols** checkbox to display symmetry planes (in 3D) and lines (in 2D) in the **Graphics** window, as specified in the Symmetry settings.

THICKNESS (2D COMPONENTS)

For 2D components, 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. Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D components.

Select the **Use manual terminal sweep** checkbox to invoke a parametric sweep over 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.

📝	For most applications it is simpler to sweep over terminals using the Stationary Source Sweep study step. See Stationary Source Sweep in the *COMSOL Multiphysics Reference Manual* and (for more details) Stationary Source Sweep in the Modeling with the AC/DC Module chapter of this manual.

⚠️	To correctly calculate the terminal charge and the corresponding capacitances, if voltage Terminal is used on boundaries between The Electrostatics Interface and The Electrostatics, Boundary Elements Interface, let the dependent variable for The Electrostatics, Boundary Elements Interface be the same as the dependent variable for The Electrostatics Interface and disable Electric Scalar-Scalar Potential Coupling. How to set the name for the dependent variable is described in the Dependent Variables section. Since the Electric Scalar-Scalar Potential Coupling is disabled, add manually a Fully Coupled node to the Stationary Solver below the Solution node in the Solver Configurations.

SYMMETRY

🔷	For 3D components, from the **Condition for the x = x₀ plane**, **Condition for the y = y₀ plane**, and **Condition for the z = z₀ plane** lists, choose **Off** (the default), **Symmetric**, or **Antisymmetric**. Then enter the value for the plane location x_0, y_0, or z_0 (SI unit: m) as required.

🔵	For 2D components, from the **Condition for the x = x₀ plane** and **Condition for the y = y₀ plane** lists, choose **Off** (the default), **Symmetric**, or **Antisymmetric**. Then enter the value for the plane location x_0 or y_0 (SI unit: m) as required.

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

	For more information about the **Far Field Approximation** settings, see Far-Field Approximation Settings in the *COMSOL Multiphysics Reference Manual*.

INFINITY CONDITION

	For 3D components, select the **Infinity condition** — **Total charge** (the default) or **Asymptotic value at infinity**. Specify the **Total charge** Q_{tot} (SI unit: C; the default value is 0 C) or **Electric potential at infinity** V_∞ (SI unit: V; the default value is 0 V). If there is an antisymmetric symmetry in the potential field, it acts as an added infinite ground plane with a fixed value of the electric potential. The value at infinity is fixed to 0 by the presence of the infinite ground plane, so for this case there is a fixed **Zero potential at infinity** condition.

	For 2D components, select the **Infinity condition** — **Total charge** (the default) or **Value at reference distance**. Specify the **Total charge** Q_{tot} (SI unit: C; the default value is 0 C) or **Electric potential at reference distance** V_{ref} (SI unit: V; the default value is 0 V). If there is an antisymmetric symmetry in the potential field, it acts as an added ground line with a fixed value of the electric potential. The value at a reference distance is fixed to 0 by the presence of the ground line, so for this case there is a fixed **Zero potential at reference distance** condition.

QUADRATURE

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

	For more information about the **Quadrature** settings, see Quadrature in the *COMSOL Multiphysics Reference Manual*.

From the **Electric potential/Surface charge density** list, choose from predefined options for the boundary element discretization order for the electric potential variable and the surface charge density variable, respectively. The predefined options represent the suitable combinations of element orders such as **Quadratic/Linear** (the default).

> For 3D components, the Line charge variable will have the same discretization order as the Electric potential variable.

The settings under **Value types when using splitting of complex variables** are important for sensitivity and optimization computations. See the description of the built-in operators fsens and fsensimag.

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.

Domain, Boundary, and Edge Nodes for the Electrostatics, Boundary Elements Interface

The Electrostatics, Boundary Elements Interface has these domain, boundary, and edge nodes available from the **Physics** ribbon (Windows users), **Physics** context menu (Mac or Linux users), or by right-clicking the physics interface in **Model Builder** to access the context menu (all users).

> In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

These nodes and subnodes are described in this section:

- Initial Values
- Charge Conservation
- Electric Potential
- Floating Potential
- Force Calculation

- Ground
- Surface Charge Density
- Terminal
- Zero Charge

The Zero Charge node is the default boundary and edge condition available for this physics interface and described for the Electrostatics interface.

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. Add more **Initial Values** nodes from the **Physics** toolbar, **Global** menu.

INITIAL VALUES

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

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 defined by the relative permittivity.

CONSTITUTIVE RELATION D-E

Enter the **Relative permittivity** ε_r (dimensionless) to describe the macroscopic properties of the medium. The default is to take the **Relative permittivity** values **From material**. For **User defined**, enter a value or expression in the field.

| ! | Notice that the boundary element method is based on the availability of an analytic Green's function for the domain. Thus, the **Relative permittivity** must evaluate to a constant value for each **Charge Conservation** node. |

Electric Potential

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

!	For more information about the Electric Potential condition, see Electric Potential in the documentation for the Electrostatics interface.

▥	For 3D components, enter the **Edge radius** (SI unit: m) for the virtual tube surrounding the edge. The default value is 0.1 m. It is generally assumed that the edge radius is small in comparison to other geometrical measures of the model, such as the edge length and the distances between different edges in the geometry.

Floating Potential

The **Floating Potential** node is used when modeling a metallic electrode at floating potential.

!	For more information about the Floating Potential condition, see Floating Potential in the documentation for the Electrostatics interface.

▥	For 3D components, enter the **Edge radius** (SI unit: m) for the virtual tube surrounding the edge. The default value is 0.1 m. It is generally assumed that the edge radius is small in comparison to other geometrical measures of the model, such as the edge length and the distances between different edges in the geometry.

Ground

The **Ground** node implements ground (zero potential) as the boundary condition $V = 0$.

!	For more information about the Ground condition, see Ground in the documentation for the Electrostatics interface.

For 3D components, enter the **Edge radius** (SI unit: m) for the virtual tube surrounding the edge. The default value is 0.1 m. It is generally assumed that the edge radius is small in comparison to other geometrical measures of the model, such as the edge length and the distances between different edges in the geometry.

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.

For more information about the Surface Charge Density condition, see Surface Charge Density in the documentation for the Electrostatics interface.

For 3D components, enter the **Edge radius** (SI unit: m) for the virtual tube surrounding the edge. The default value is 0.1 m. It is generally assumed that the edge radius is small in comparison to other geometrical measures of the model, such as the edge length and the distances between different edges in the geometry. Enable **Compensate for tube volume** to allow for a potential gradient across the cross section of the virtual tube. This can be useful for establishing a zero surface charge density condition on the virtual tube periphery.

Terminal

The **Terminal** node provides a boundary or domain condition for connection to external circuits, to transmission lines, or with a specified voltage or charge.

For more information about the Terminal condition, see Terminal in the documentation for the Electrostatics interface.

For 3D components, enter the **Edge radius** (SI unit: m) for the virtual tube surrounding the edge. The default value is 0.1 m. It is generally assumed that the edge radius is small in comparison to other geometrical measures of the model, such as the edge length and the distances between different edges in the geometry.

To correctly calculate the terminal charge and the corresponding capacitances, if voltage Terminal is used on boundaries between The Electrostatics Interface and The Electrostatics, Boundary Elements Interface, let the dependent variable for The Electrostatics, Boundary Elements Interface be the same as the dependent variable for The Electrostatics Interface and disable Electric Scalar-Scalar Potential Coupling. How to set the name for the dependent variable is described in the Dependent Variables section. Since the Electric Scalar-Scalar Potential Coupling is disabled, add manually a Fully Coupled node to the Stationary Solver below the Solution node in the Solver Configurations.

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 is the default boundary condition on exterior boundaries and on edges embedded within a Charge Conservation feature.

Electric Scalar-Scalar Potential Coupling

The **Electric Scalar-Scalar Potential Coupling** multiphysics node assures continuity of the electric potential across boundaries between The Electrostatics Interface and The Electrostatics, Boundary Elements Interface. The **Electric Scalar-Scalar Potential Coupling** node is available from the **Multiphysics** menu in the **Physics** toolbar or by right-

clicking the **Multiphysics Couplings** node in **Model Builder**, if both The Electrostatics Interface and The Electrostatics, Boundary Elements Interface are available.

	The Electrostatics Interface and The Electrostatics, Boundary Elements Interface can also be coupled by using the same name for the dependent variable for both interfaces. Then **Electric Scalar-Scalar Potential Coupling** is not needed. How to set the name for the dependent variable is described in the Dependent Variables section.

	When using The Electrostatics, Boundary Elements Interface with iterative solvers, **Right Preconditioning** is preferred. Thus, it is advised to check that **Right Preconditioning** is used when combining the **Electric Scalar-Scalar Potential Coupling** with The Electrostatics, Boundary Elements Interface and other physics interfaces. How to define **Right Preconditioning** is described in Iterative in the *COMSOL Multiphysics Reference Manual*.

BOUNDARY SELECTION

Select **Manual** or **All boundaries** from the **Selection** list. Make additional edits to the list of boundary entity numbers using the Selection toolbar buttons. When **All boundaries** is selected from the **Selection** list, the boundaries exterior to the Electrostatics interface that intersect the exterior boundaries to the Electrostatics, Boundary Elements interface are available in the boundary entity number list.

COUPLED INTERFACES

Select **Electrostatics, Boundary Elements** as **Primary interface** and **Electrostatics** as **Secondary interface**.

The Electric Currents Interface

The **Electric Currents (ec)** interface ( ), under the **AC/DC** > **Electric Fields and Currents** branch when adding a physics interface, is used to compute electric field, current, and potential distributions in conducting media under conditions where inductive effects are negligible; that is, when the skin depth is much larger than the studied device.

Depending on the licensed products, stationary, frequency-domain, small-signal analysis, and time-domain modeling are supported in all space dimensions. In the time and frequency domains, capacitive effects are also accounted for.

The physics interface solves a current conservation equation based on Ohm's law using the scalar electric potential as the dependent variable. The interface is typically used to model good conductors for which Ohm's law applies. It is not suitable for modeling poor conductors like air and insulators.

Current Conservation is the main node, which adds the equation for the electric potential and provides a Settings window for defining the electric conductivity as well as the constitutive relation for the electric displacement field and its associated material properties, such as the relative permittivity.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Current Conservation**, **Electric Insulation** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and current sources. You can also right-click **Electric Currents** to select physics features from the context menu.

Physics-Controlled Mesh
The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the **Sequence type** is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the checkbox in the **Use** column on the same table row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

Information from the physics, such as the presence of an infinite elements domain or periodic condition, will be used to automatically set up an appropriate meshing sequence.

> | ! | In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is ec.

CROSS-SECTION AREA (1D)

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

THICKNESS (2D)

Enter a default value for the **Out-of-plane thickness** d (SI unit: m) (see Equation 2-20). The default value of 1 m 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 components. See also Change Thickness (Out-of-Plane) (described for the Electrostatics interface).

MANUAL TERMINAL SWEEP SETTINGS

Enter a **Reference impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω.

Select the **Use manual terminal sweep** checkbox 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 a **Parameter format (value pairs)** for the Touchstone export — **Magnitude and angle (MA)** (the default), **Magnitude in dB and angle (DB)**, or **Real and imaginary parts (RI)**. Select an option from the **If file exists** list — **Overwrite** or **Create new**. Create new is useful when the model is solved multiple times with different settings. Select a **Parameter to export** — **Z** (the default), **Y**, or **S**.

📋	For most applications it is simpler to sweep over terminals using the Stationary Source Sweep study step. See Stationary Source Sweep in the *COMSOL Multiphysics Reference Manual* and (for more details) Stationary Source Sweep in the *AC/DC Module User's Guide*.

DEPENDENT VARIABLES

The dependent variable is the **Electric potential** V. You can change its name, which changes both the field name and the variable name. If the new name coincides with the name of another electric potential field in the model, the physics interfaces share degrees of freedom. The new name must not coincide with the name of a field of another type or with a component name belonging to some other field.

DISCRETIZATION

Select the shape order for the **Electric potential** dependent variable — **Linear**, **Quadratic** (the default), **Cubic**, **Quartic**, or **Quintic**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

🔍	• Computing Lumped Parameters • Domain, Boundary, Edge, Point, and Pair Nodes for the Electric Currents Interface • Theory for the Electric Currents Interface

▦	*Pacemaker Electrode*: Application Library path **COMSOL_Multiphysics/ Electromagnetics/pacemaker_electrode**

Domain, Boundary, Edge, Point, and Pair Nodes for the Electric Currents Interface

The Electric Currents interface has these domain, boundary, edge, point, and pair nodes available from the **Physics** ribbon toolbar (Windows users) or **Physics** context menu (Mac or Linux users). You can also right-click to access the context menu (all users).

> In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

ABOUT THE BOUNDARY CONDITIONS

The exterior and interior boundary conditions listed in Table 4-1 are available. The relevant physics 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.

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

- Boundary Current Source
- Contact Impedance
- Current Conservation
- Current Source
- Dispersion
- Distributed Impedance
- Electric Insulation
- Electric Point Dipole
- Electric Point Dipole (on Axis)
- Electric Shielding
- Electrical Contact

- External Current Density
- Floating Potential
- Initial Values
- Line Current Source
- Line Current Source (on Axis)
- Normal Current DensityPoint Current Source
- Sector Symmetry
- Symmetry Plane (for Electric Field)
- Terminal

These nodes are described for the Electrostatics interface:

- Archie's Law
- Change Cross Section
- Change Thickness (Out-of-Plane)
- Effective Medium

- Electric Potential
- Force Calculation
- Ground
- Periodic Condition

⚠	In the *COMSOL Multiphysics Reference Manual*, see Table 2-4 for links to common sections and Table 2-5 for common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

For axisymmetric components, 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.

Table 4-1 lists the interior and exterior boundary conditions available with this physics 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.

NODE	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
Symmetry Plane		x	not applicable
Terminal	x	x	pairs, domains

Current Conservation

The **Current Conservation** node adds the continuity equation for the electric potential and provides an interface for defining the electric conductivity as well as the constitutive relation and the relative permittivity for the displacement current.

MATERIAL TYPE

The **Material type** setting decides how materials behave and how material properties are interpreted when the mesh is deformed. Select **Solid** for materials whose properties change as functions of material strain, material orientation, and other variables evaluated in a material reference configuration (material frame). Select **Nonsolid** for materials whose properties are defined only as functions of the current local state at each point in the spatial frame, and for which no unique material reference

configuration can be defined. Select **From material** to pick up the corresponding setting from the domain material on each domain.

CONSTITUTIVE RELATION JC-E

By default, the **Electric conductivity** σ (SI unit: S/m) for the media is defined **From material**. Or select **User defined**, **Effective medium**, **Archie's Law**, **Hall effect**, or **Linearized resistivity**. When **Effective medium** is selected, either right-click Current Conservation or in the **Physics** toolbar, **Attributes** menu click to add an Effective Medium subnode. When **Archie's law** is selected, do the same to add an Archie's Law subnode.

User Defined

For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** depending on the characteristics of the electric conductivity, and then enter values or expressions for the electric conductivity σ in the field or matrix. The default is 0 S/m. If 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:

$$\sigma = \frac{1}{\rho_0(1 + \alpha(T - T_{\text{ref}}))}$$

where ρ_0 is the resistivity at the reference temperature T_{ref}, and α is the temperature coefficient of resistance, which describes how the resistivity varies with temperature.

The default **Reference resistivity** ρ_0 (SI unit: $\Omega \cdot$m), **Reference temperature** T_{ref} (SI unit: K), and **Resistivity temperature coefficient** α (SI unit: 1/K) are taken **From material**, which means that the values are taken from the domain (or boundary) material. 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 is in the **Model Inputs** section.

	Only certain material models support the **Linearized resistivity**. See AC/ DC Material Library in the *COMSOL Multiphysics Reference Manual*.

To specify other values for any of these properties, select **User defined** from the list and then enter a value or expression for each. The default values are:

- 1 Ω·m for the Reference resistivity
- 273.15 K for the Reference temperature, and
- 0 1/K for the Resistivity temperature coefficient

Hall Effect

Select **Hall Effect** to incorporate the effect of a magnetic field on the constitutive relation. A Hall current term $\sigma R_H \mathbf{J}_c \times \mathbf{B}$ is added to the total conduction current $\mathbf{J}_c$, where R_H is the Hall coefficient, and $\mathbf{B}$ is the magnetic flux density. This current component is incorporated using an anisotropic modified electric conductivity tensor.

The specified **Electric conductivity** is isotropic and can be defined **From material** or **User defined**. The **Hall coefficient** R_H (SI unit: $m^3/(s \cdot A)$) is defined macroscopically as the inverse of the carrier charge density $R_H = 1/(nq)$, with n the carrier density, and q the carrier charge, with the value set by user input. The default magnetic flux density is **User defined**, but can also be defined from **Common model input** or from an appropriate magnetic fields interface.

Not that the **Hall Effect** model is based on the assumption that changes in the magnetic field are relatively small and do not produce significant induced currents. Therefore it is only suitable for stationary studies. In cases where the induced electric field is negligible, the stationary formulation can be used in time-dependent studies by setting the **Equation form** to **Stationary**, instead of the default **Study controlled**.

CONSTITUTIVE RELATION D-E

Select a **Dielectric model** 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. For a description of the constitutive relations **Relative permittivity**, **Polarization**, **Remanent electric displacement** and **Dielectric losses**, see **Constitutive Relation D-E** as described for the Charge Conservation node for the Electrostatics interface. The constitutive relations specific to Electric Currents are:

- '**Loss tangent, loss angle**: uses the constitutive relation $\mathbf{D} = \varepsilon_0 \varepsilon'(1 - j\tan\delta)\mathbf{E}$. Specify the **Relative permittivity (real part)** ε' (dimensionless) and **Loss angle** δ (SI unit: rad).
- **Loss tangent, dissipation factor**: uses the constitutive relation $\mathbf{D} = \varepsilon_0 \varepsilon'(1 - j\tan\delta)\mathbf{E}$. Specify the **Relative permittivity (real part)** ε' (dimensionless) and the **Dissipation factor** $\tan\delta$ (dimensionless).

- **Dispersion** to use the constitutive relation $\mathbf{D} = \varepsilon_0\mathbf{E} + \mathbf{P}(\mathbf{E}, \varepsilon_{\mathrm{rS}})$, where the polarization vector is calculated from the electric field using the dielectric dispersion model. This option is available only when the **Material type** is set to **Solid**. You enter the **Relative permittivity** $\varepsilon_{\mathrm{rS}}$ (dimensionless) values **From material**. For **User defined**, select **Isotropic**, **Diagonal**, or **Symmetric** and enter values or expressions in the field or matrix. This value of electric permittivity will be used in stationary study, for which the polarization vector is calculated as $\mathbf{P} = \varepsilon_0(\varepsilon_{\mathrm{rS}} - \mathrm{I})\mathbf{E}.$

🔍	Once this option is selected, a subnode **Dispersion** will become available under the **Current Conservation** node. At that subnode, you can select the dispersion model, enter the corresponding parameters, and choose how the relative permittivity input on the parent node will be interpreted in Eigenfrequency, Frequency Domain, and Time Dependent studies.

Dispersion

The **Dispersion** subnode to **Current Conservation** allows you to model possible dielectric losses in the material. The effect can be accounted for in Eigenfrequency, Frequency Domain and Time Dependent study.

Under the **Material model**, you can select either **Debye** (default) or **Multipole Debye** dispersion option.

For a single pole **Debye** dispersion choice, you can enter the **Relaxation time** and **Relative permittivity contribution**. In the Thermal Effects tab, you can select the **Shift function** type, which will be used to compute the effective relaxation time accounting for the temperature effects.

For the **Multipole Debye** dispersion, you can select two options to enter the **Relaxation Data**.

- **Relaxation time and Relative permittivity change** choice will provide an input table with dynamically changed size, where you can enter any number of **Relaxation time** and the corresponding **Relative permittivity contribution** values. In the Thermal Effects tab, you can select the **Shift function** type, which will be used to compute the effective relaxation times accounting for the temperature effects.
- **Constant Loss Tangent** option. In this case, you enter the **Loss Tangent** $\eta(f_c)$ together with the **Center frequency** f_c. You also specify the model **Bandwidth (decades)** that defines a frequency interval centered at f_c, in which the loss tangent will be approximately constant and equal to $\eta(f_c)$. You can also select the **Accuracy** for the

approximation to be either **Normal** (default) or **High**. The software will automatically deduce the necessary number of Debye poles together with the values of the corresponding relaxation times and relative permittivity contributions, which will be used in computations to maintain the requested bandwidth and accuracy.

🔍	Dispersion Theory

⚠️	In Eigenfrequency study and Time Dependent study, each pole will require one extra vector degree of freedom (domain variable) to represent the corresponding contribution to the polarization. Thus, using many poles in the relaxation data input table, or setting the **Accuracy** to **High** accuracy in case of **Constant Loss Tangent**, can lead to significant computation costs for larger models.

For all cases, you can specify how the relative permittivity input, ε_{rS}, on the parent node should be interpreted by selecting the **Static response** (the default value is **High frequency limit**). This setting will have effect in Eigenfrequency, Frequency Domain, and Time Dependent study types.

THERMAL EFFECTS

Dispersion properties depend on the temperature. For many materials, a change in the temperature can be transformed directly into a change in the time scale. Thus, the relaxation time is modified to $a_T(T)\tau_m$, where $a_T(T)$ is a shift function.

Select a Shift function — **None**, **Vogel-Fulcher**, **Arrhenius**, **Williams-Landel-Ferry**, **Tool-Narayanaswamy-Moynihan**, or **User defined**.

- When the default, **None**, is kept, the shift function $a_T(T)$ is set to unity and the relaxation time is not modified.

- For **Vogel-Fulcher** enter values or expressions for these properties:

    - **Reference temperature** T_0. The default is 230 K.

    - **Activation energy** Q. The default is 8000 J/mol.

- For **Arrhenius** enter values or expressions for these properties:

    - **Reference temperature** T_0. The default is 293.15 K.

    - **Activation energy** Q.

- For **Williams-Landel-Ferry** enter values or expressions for these properties:

  - **Reference temperature** T_{WLF} The default is 293.15 K.

  - **WLF constant 1** $C_{1\mathrm{WLF}}$. The default is 17.44.

  - **WLF constant 2** $C_{2\mathrm{WLF}}$. The default is 51.6 K.

- For **Tool-Narayanaswamy-Moynihan** enter values or expressions for these properties:

  - **Reference temperature** T_0. The default is 293.15 K.

  - **Activation energy** Q.

  - **Activation energy fraction** χ $(0 < \chi < 1)$.

  - **Structural relaxation times** λ_0.

  - **Fictive temperature weights** w.

- For **User defined** enter a value or expression for the shift function a_{T}.

DISCRETIZATION

To display this section, click the **Show More Options** button (👁) and select **Discretization** from the **Show More Options** dialog. Select the element order from the list box for the auxiliary electric field vector variables $\mathbf{e}_n$.

You can change the solution algorithm by using the checkbox **Use local time integration** (checked by default). When unchecked, the solution of the equations for the auxiliary dependent variables in Time Dependent study will be governed by the time stepper algorithm used in the solver.

LOCATION IN USER INTERFACE

Context Menus
Electric Currents > **Current Conservation** > **Dispersion**

Ribbon
Physics tab with **Current Conservation** node selected in the model tree:

Attributes > **Dispersion**

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, from the **Physics** toolbar, add other

nodes that implement, for example, boundary conditions and current sources. Add more **Initial Values** nodes from the **Physics** toolbar.

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.

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 physics interface defines.

The external current density does not contribute to the losses (due to Joule heating), since there is no electric field associated with it. To include the contribution to the losses from the external current density, select the **Add contribution of the external current density to the losses** checkbox. Then select an option from the **External losses** list — **From domain conductivity** (the default) or **User defined**. If **From domain conductivity** is selected, the heat source is computed using the conductivity specified in the material model feature (such as **Current Conservation**) that is applied in the domain. For **User defined**, enter a value for Q_e (SI unit: W/m^3) to specify a user-defined heat source.

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

EXTERNAL CURRENT DENSITY

Based on space dimension, enter the coordinates (**x**, **y**, and **z** for 3D components, for example) of the **External current density** $\mathbf{J}_e$ (SI unit: A/m^2). The defaults are 0 A/m^2.

Current Source

The **Current Source** node adds a distributed current source Q_j in the equation that the physics interface defines. Use this node with caution as it can violate the current conservation law that is inherent in Maxwell–Ampère's law.

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual.*

CURRENT SOURCE

Enter a value or expression for the **Current source** Q_j (SI unit: A/m^3). The default is 0 A/m^3.

Electric Insulation

The **Electric Insulation** node, which is the default boundary condition, 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.

Electric insulation as the default boundary condition is not applicable to interior boundaries. To add electric insulation to an interior boundary, add an **Electric Insulation** node in addition to the one that represents the default boundary condition.

Floating Potential

The **Floating Potential** node is used when modeling a metallic electrode at floating potential. It applies a constant voltage V_0 on the boundary (for domain features, this is the boundary enclosing the selected domain), such that the total normal electric current density $\mathbf{J}$ equals a specific current I_0:

$$V = V_0$$

$$\int_{\partial\Omega} (-\mathbf{n} \cdot \mathbf{J})dS = I_0,$$

where Ω represents the boundary and $\mathbf{n}$ refers to the surface normal. The constant boundary voltage implies the tangential electric field equals zero; the electric field will be perpendicular to the boundary:

$$-\nabla_t V = \mathbf{E}_t = 0$$
$$\mathbf{n} \times \mathbf{E} = 0.$$

In case of $I_0 = 0$ (the default case), the boundary will behave as an unconnected perfect conductor (a floating equipotential). This is a good approximation when the conductivity of the electrode is many orders of magnitude larger than that of the surrounding medium. Although locally the current density may vary, the total current entering or leaving the boundary equals zero.

If the floating potential touches a point, boundary, or domain feature that is not floating (a **Terminal** or **Ground** feature), the floating potential will acquire that feature's potential. If the floating potential is set to a certain current, or connected to a circuit, it behaves like a terminal.

The **Harmonic Perturbation** subnode (it is of the exclusive type) is available from the context menu (right-click the parent node) or in the **Physics** toolbar, click the **Attributes** menu and select **Harmonic Perturbation**. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

FLOATING POTENTIAL

The Floating potential identifier text area shows the unique identifier for the floating potential feature. It is used to identify the global variables created by the node, such as the voltage. The **Floating potential group** checkbox in the **Settings** window for **Floating Potential** controls how potentials are assigned to boundary segments in the feature's boundary selection. If this checkbox is not selected, a single potential is applied to all boundaries in the selection. If the checkbox is selected, each group of contiguous boundaries in the selection is given a unique potential. This simplifies the setup of models with many floating electrodes. The values of the potential at each group of boundaries are then made available in postprocessing, collected in a vector variable.

The following options are not available if the **Floating potential group** checkbox is selected. Select a specification for the **Floating potential current** — choose **User defined** (the default) to specify a total **Current** I_0 (SI unit: A) that flows from the electrode. The default is 0 A, corresponding to an unconnected electrode. Select **Circuit** to connect the floating potential to an Electrical Circuit interface.

Initial values
The initial values subsection is used to provide initial values for the internal state variables in the Floating Potential. These settings can be useful to provide an initial guess to the nonlinear solver or the initial value in a time-dependent study when the

initial values of the other dependent variables (such as the electric potential) are computed from a separate physics or study. In most cases, the default values are appropriate.

Enter an **Initial value for voltage** V_{init} (SI unit: V). The default is 0 V.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

🔍	Settings and Properties Windows for Feature Nodes in the *COMSOL Multiphysics Reference Manual*

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. Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or click **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

BOUNDARY CURRENT SOURCE

Enter a value or expression for the **Boundary current source** Q_j (SI unit: A/m^2). The default is 0 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. Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

NORMAL CURRENT DENSITY

Select a **Type** — **Inward current density** (the default) or **Current density**.

- For **Inward current density** enter a value or expression for the **Normal current density** J_n (SI unit: A/m^2). Use a positive value for an inward current flow or a negative value for an outward current flow. The default is 0 A/m^2.

- For **Current density** enter values or expressions for the components of the **Current density $\mathbf{J}_0$** (SI unit: A/m^2). The defaults are 0 A/m^2.

Distributed Impedance

The **Distributed Impedance** node adds a distributed impedance boundary condition to a model.

The **Harmonic Perturbation** subnode (it is of the exclusive type) is available from the context menu (right-click the parent node) or in the **Physics** toolbar, click the **Attributes** menu and select **Harmonic Perturbation**. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

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 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:

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = \frac{(\sigma + j\omega\varepsilon_0\varepsilon_r)}{d_s}(V - V_{\mathrm{ref}})$$

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = \left(\frac{1}{r_s} + j\omega C_s\right)(V - V_{\mathrm{ref}})$$

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = \frac{1}{d_L}\left(\sigma(V - V_{\mathrm{ref}}) + \varepsilon_0\varepsilon_r\frac{\partial}{\partial t}(V - V_{\mathrm{ref}})\right)$$

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = \left(\frac{1}{\rho_s}(V - V_{\mathrm{ref}}) + C_s\frac{\partial}{\partial t}(V - V_{\mathrm{ref}})\right)$$

DISTRIBUTED IMPEDANCE

Enter the reference potential V_{ref} (SI unit: V). The default is 0 V.

Select a potentially complex-valued **Layer specification** — **Thin layer** (the default) or **Surface impedance**.

- For **Thin layer**, enter values or expressions for the:
  - **Surface thickness** d_s (SI unit: m). The default is $5 \cdot 10^{-3}$ m (5 mm).
  - **Electric conductivity** σ (SI unit: S/m) and **Relative permittivity** ε_r (dimensionless). The defaults take values **From material**. For **User defined**, enter different values or expressions. The default electric conductivity is $1 \cdot 10^{-2}$ S/m and the default relative permittivity is 1.

- For **Surface impedance**, enter values or expressions for the **Surface resistance** ρ_s (SI unit: $\Omega \cdot \mathrm{m}^2$) and the **Surface capacitance** C_s (SI unit: F/m^2). The default surface impedance is $1 \cdot 10^{-8}$ $\Omega \cdot \mathrm{m}^2$ and the default surface capacitance is 0 F/m^2.

Terminal

The **Terminal** node provides a boundary or domain 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.

> ⚠️ In most cases, it is not recommended to include the interior of good conductors in the simulation. When using the **Terminal** on the domain level, the interior unknowns for the electric potential are replaced by a single constant, thus avoiding numerical problems arising from a very high material contrast. If one instead uses the **Terminal** on the boundary level, it is recommended to exclude the interior of the associated conductor from the domain selection for the main Electric Currents node.

The **Terminal** node also enables the computation of the lumped parameter of the system, such as the impedance.

The **Harmonic Perturbation** subnode (it is of the exclusive type) is available from the context menu (right-click the parent node) or in the **Physics** toolbar, click the **Attributes** menu and select **Harmonic Perturbation**. The **Harmonic Perturbation** subnode only has effect if **Current** or **Voltage** are selected as terminal type. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

An operation named as **Split by Connectivity** (✏️) is available by right-clicking the parent node. The operation is useful when there are several terminal domains or boundaries that are not geometrically connected. Clicking the operation node will automatically generate several **Terminal** nodes where each selection is geometrically connected.

TERMINAL

Specify the terminal's properties. The **Terminal name** identifies the terminal and the variables it creates. The name should be numeric for sweeps to work properly.

Select a **Terminal type** — **Current** (the default), **Voltage**, **Circuit**, **Power**, **Terminated (power)**, and **Terminated (voltage)**.

!	The **Circuit** and **Power** types cannot be used together with a terminal sweep.

Select:

- **Current** to specify a total current flowing from the terminal. Enter the current I_0 (SI unit: A). The default is 0 A for an open circuit.

- **Voltage** to specify a fixed electric potential at the terminal. Enter the value of the electric potential V_0 (SI unit: V). The default is 1 V.

- **Circuit** to specify a terminal connected to an Electrical Circuit interface that models an external circuit.

- **Power** to specify a fixed electric power for the terminal. The power is computed as the product of the terminal current and voltage and, in the frequency domain, is the cycle-averaged power. When **Power** is selected:

    - Enter a **Terminal power** P_0 (SI unit: W) to specify the input power at this terminal. The default is 1 W.

- **Terminated (power)** to connect the terminal to an impedance that might represent a load or a transmission line. When **Terminated (power)** 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 terminal sweep. The default is 1 W.

    - Select an option from the **Characteristic impedance from** list to define the value of the impedance — **Physics interface** (the default) or **User defined**. For **Physics interface** the **Reference impedance** Z_{ref} defined in the **Settings** window for the

physics interface under Sweep Settings is used. For **User defined** enter a specific impedance Z_{ref} (SI unit: Ω) for this terminal. The default is 50 Ω.

- **Terminated (voltage)** to connect the terminal to an impedance that might represent a load or a transmission line. The **Terminated (voltage)** option is essentially the same as **Terminated (power)**, with the power defined as: $P_0 = 0{,}5 \cdot |V_0|^2 / Z_{ref}$.

!	Using Power excitation causes the equation system to become nonlinear. COMSOL Multiphysics automatically adjusts the solver settings when Power is selected.
	Be aware that, in general, the values of the voltage and the current might not uniquely be determined by this constraint and may depend on the initial value provided for the current and voltage variables. 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 Manual* for a possible solution to this problem.

Initial Values

The initial values subsection is used to provide initial values for the internal state variables in the Terminal. These settings can be useful to provide an initial guess to the nonlinear solver or the initial value in a time-dependent study when the initial values of the other dependent variables (such as the electric potential) are computed from a separate physics or study. In most cases, the default values are appropriate.

The **Initial value for voltage** is available for all Terminal types except Voltage. Enter an **Initial value for voltage** V_{init} (SI unit: V). The default is 0 V.

The **Initial value for current** is available for Power only. In this case, it provides an initial guess for the nonlinear solver, and should not be set to zero. The initial value provided may affect the solution. Enter an **Initial value for current** I_{init} (SI unit: A). The default is 1 mA.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

🔍	Computing Lumped Parameters

Electric Shielding

The **Electric Shielding** node provides an electric shielding boundary condition. Use this node to model a thin layer of a highly conductive medium that shields the electric field. The sheet has the electric 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. The **Electric Field** and **Conduction Current** sections are the same as for Current Conservation.

THICKNESS

Enter a value or expression for the **Surface thickness** d_s (SI unit: m). The default is 1 cm.

▥	*Electric Shielding*: Application Library path **ACDC_Module/Introductory_Electric_Currents/electric_shielding**

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 using a **Pair Contact Impedance** node. The feature allows specifying the contact impedance either by entering the properties of the material together with the layer thickness, or by entering the impedance properties of the thin layer directly.

The feature, in stationary studies, applies the following conditions that relate the normal electric current density with the jump in the electric potential:

$$\mathbf{n} \cdot \mathbf{J}_1 = \frac{\sigma}{d_s}(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \frac{\sigma}{d_s}(V_2 - V_1)$$

$$\mathbf{n} \cdot \mathbf{J}_1 = \frac{1}{\rho_s}(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \frac{1}{\rho_s}(V_2 - V_1)$$

The first two equations refer to a layer impedance specified using the bulk material conductivity σ_s and the layer thickness d_s, while the last two equations refer to the case in which the surface resistance ρ_s is specified. The indices 1 and 2 refer to the two sides of the boundary. These parameters work the same as with Distributed Impedance.

For the frequency domain and Time Dependent study types, this boundary condition is slightly more sophisticated and accounts also for capacitive coupling. The bulk material relative permittivity ε_r or the layer capacitance C_s can be specified.

The equations used in Frequency Domain studies are:

$$\mathbf{n} \cdot \mathbf{J}_1 = \frac{(\sigma + j\omega\varepsilon_0\varepsilon_r)}{d_s}(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \frac{(\sigma + j\omega\varepsilon_0\varepsilon_r)}{d_s}(V_2 - V_1)$$

$$\mathbf{n} \cdot \mathbf{J}_1 = \left(\frac{1}{\rho_s} + j\omega C_s\right)(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \left(\frac{1}{\rho_s} + j\omega C_s\right)(V_2 - V_1)$$

In Time Dependent studies:

$$\mathbf{n} \cdot \mathbf{J}_1 = \frac{1}{d_s}\left(\sigma(V_1 - V_2) + \varepsilon_0\varepsilon_r\frac{\partial}{\partial t}(V_1 - V_2)\right)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \frac{1}{d_s}\left(\sigma(V_2 - V_1) + \varepsilon_0\varepsilon_r\frac{\partial}{\partial t}(V_2 - V_1)\right)$$

$$\mathbf{n} \cdot \mathbf{J}_1 = \left(\frac{1}{\rho_s}(V_1 - V_2) + C_s\frac{\partial}{\partial t}(V_1 - V_2)\right)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \left(\frac{1}{\rho_s}(V_2 - V_1) + C_s\frac{\partial}{\partial t}(V_2 - V_1)\right)$$

Select a potentially complex-valued **Layer specification** — **Thin layer** (the default) or **Surface impedance**.

- For **Thin layer**, enter values or expressions for the:

  - **Surface thickness** d_s (SI unit: m). The default is $5 \cdot 10^{-3}$ m (5 mm).

  - **Electric conductivity** σ (SI unit: S/m) and **Relative permittivity** ε_r (dimensionless). The defaults take values **From material**. For **User defined**, enter different values or expressions. The default electric conductivity is $1 \cdot 10^{-2}$ S/m and the default relative permittivity is 1.

- For **Surface impedance**, enter values or expressions for the **Surface resistance** ρ_s (SI unit: $\Omega \cdot m^2$) and the **Surface capacitance** C_s (SI unit: F/m^2). The default surface impedance is $1 \cdot 10^{-8}$ $\Omega \cdot m^2$ and the default surface capacitance is 0 F/m^2.

▦	*Thin-Film Resistance*: Application Library path **COMSOL_Multiphysics/ Electromagnetics/thin_film_resistance**

Electrical Contact

The **Electrical Contact** node defines correlations for the conductance h at the interface of two bodies in an imperfect contact due to the surface roughness. It can be added to pairs by selecting **Pair Electrical Contact** from the **Pairs** menu.

The conductance h is involved in the current flow across the surfaces in contact according to:

$$\mathbf{n} \cdot J_1 = -h_c(V_1 - V_2)$$

$$\mathbf{n} \cdot J_2 = -h_c(V_2 - V_1)$$

where indices 1 and 2 refer to the two sides of the boundary.

ELECTRICAL CONTACT

Select the **Constriction conductance** — **Cooper-Mikic-Yovanovich correlation** (the default), **Mikic elastic correlation**, or **User defined**. For **User defined** enter a value or expression for h_c (SI unit: S/m^2). The default is 0 S/m^2.

CONTACT SURFACE PROPERTIES

This section displays if **Cooper-Mikic-Yovanovich correlation** or **Mikic elastic correlation** are chosen under **Contact**.

Enter values for the:

- **Surface roughness, asperities average height** σ_{asp} (SI unit: m). The default is 1 μm.
- **Surface roughness, asperities average slope** m_{asp} (dimensionless). The default is 0.4.
- **Contact pressure** p (SI unit: Pa). The default is 0 Pa.

For **Cooper-Mikic-Yovanovich correlation** select a **Hardness definition** — **Microhardness** (the default), **Vickers hardness**, or **Brinell hardness**.

- For **Microhardness** enter a value for H_c (SI unit: Pa). The default is 3 GPa.
- For **Vickers hardness** enter a value for the **Vickers correlation coefficient** c_1 (SI unit: Pa) and **Vickers size index** c_2 (dimensionless). The defaults are 5 GPa and −0.1, respectively.
- For **Brinell hardness** enter a value for H_B (SI unit: Pa). The default is 3 GPa. H_B should be between 1.30 and 7.60 GPa.

For **Mikic elastic correlation** select the **Contact interface Young's modulus** $E_{contact}$ — **Weighted harmonic mean** (the default) or **User defined**.

- For **Weighted harmonic mean**, enter values or expressions for the **Young's modulus, upside**, E_u (SI unit Pa), the **Young's modulus, downside**, E_d (SI unit Pa), the **Poisson's ratio, upside**, ν_u (dimensionless), and the **Poisson's ratio, downside**, ν_d (dimensionless). If this node is selected from the **Pairs** menu, enter instead values or expressions for the **Young's modulus, source**, E_{src} (SI unit Pa), the **Young's modulus, destination**, E_{dst} (SI unit Pa), the **Poisson's ratio, source**, ν_{src} (dimensionless), and the **Poisson's ratio, destination**, ν_{dst} (dimensionless). The defaults are 0.
- For **User defined** enter another value or expression for the **Contact interface Young's modulus** $E_{contact}$ (SI unit: Pa). The default is 1 GPa.

🔍   Theory for Electrical Contact

Sector Symmetry

Select **Sector Symmetry** at interfaces between rotating objects where sector symmetry is used. It is only available for pairs. A default subnode is added. Right-click to select

additional features from the **Fallback Features** submenu. In 2D, this feature assumes rotation around the origin.

	This feature is always used in conjunction with a Periodic Condition on adjacent radial sector boundaries. Note that the same periodic condition feature cannot be used on both sides of where the sector symmetry boundaries connect with the periodic boundaries. At least two periodic condition features are required for the model to compute correctly.

PAIR SELECTION

When using nonconforming meshes on the source and destination of a pair, for numerical stability, a finer mesh should be applied on the destination side for any pair with a condition that imposes a coupling or a constraint across the pair. The sector symmetry feature falls into this category.

SECTOR SETTINGS

Enter the **Number of sectors (<50)** n_sect. The default is 2.

Select a **Type of periodicity** — **Continuity** (the default) or **Antiperiodicity**.

Based on space dimension, enter values or expressions in the table for the **Axis of rotation** $\mathbf{a}_\text{rot}$.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog. The constraint setting can be set to **Pointwise constraints** (default), or **Weak constraints**. Additionally, the **Nitsche constraints** option is available for the scalar potential in the **Electric Currents**, **Magnetic Field, No Currents**, and **Rotating Machinery, Magnetic** interfaces. The displacement current term is excluded from the **Nitsche constraints** in the **Electric Currents interface** and is therefore valid in the quasistationary approximation.

	The **Nitsche constraints** option assumes the quasistatic approximation and is applicable only for pairs adjacent to domains with small or no displacement currents or induced eddy currents normal to the pair boundary. In addition, nonlinear material models in the neighboring domains are not supported.

Line Current Source

The **Line Current Source** node adds a line source to edges in 3D components and to points in 2D and 2D axisymmetric components. The line source represents electric current per unit length.

EDGE OR POINT SELECTION

⚠	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. The default is 0 A/m.

🔍	Line Current Source (on Axis) for 2D axisymmetric components.

Line Current Source (on Axis)

The **Line Current Source (on Axis)** node adds a line source to boundaries in 2D axisymmetric components. The line source represents electric current per unit length.

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.

🔍	Line Current Source

Point Current Source

The **Point Current Source** node adds a point source and represents an electric current flowing out of the point. Add point sources to 3D components from the **Points** menu. Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** in the **Physics** toolbar. For more information see

Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

POINT SELECTION

> ⚠ 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.

> 🔍
> - Line Current Source for 2D components
> - Point Current Source (on Axis) for 2D axisymmetric components

Point Current Source (on Axis)

The **Point Current Source (on Axis)** node adds a point source and represents an electric current flowing out of the point in 2D axisymmetric components.

POINT SELECTION

> ⚠ 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.

> 🔍
> - Point Current Source for 3D components
> - Line Current Source for 2D components

Electric Point Dipole

The **Electric Point Dipole** node, available for 2D and 3D components, 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.

DIPOLE SPECIFICATION
Select a **Dipole specification** — **Magnitude and direction** (the default) or **Dipole moment**.

DIPOLE PARAMETERS
Under **Dipole Specification**:

- For **Magnitude and direction** enter coordinates for the **Electric current dipole moment direction** $\mathbf{n}_p$ and the **Electric current dipole moment, magnitude** p (SI unit: A·m).
- For **Dipole moment** enter the components of the **Electric current dipole moment** $\mathbf{p}$ (SI unit: A·m).

Q	Electric Point Dipole (on Axis)

Electric Point Dipole (on Axis)

The **Electric Point Dipole (on Axis)**, available for 2D axisymmetric components, 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.

ELECTRIC POINT DIPOLE (ON AXIS)
Enter the **Electric current dipole moment in z direction** p_z (SI unit: A·m).

Q	Electric Point Dipole

The Electric Currents in Shells Interface

The **Electric Currents in Shells (ecis)** interface ( ), found under the **AC/DC** > **Electric Fields and Currents** branch when adding a physics interface, is used to compute electric fields, currents, and potential distributions in thin conducting layered shells under conditions where inductive effects are negligible; that is, when the skin depth is much larger than the studied device. It supports stationary, time-dependent, and frequency-domain modeling on faces in 3D.

The electric potential can be located either on the selected surface or in a product space spanned by the surface and the physical thickness of the shell (using an extra dimension). For conductive shells, the physics interface solves a current conservation equation based on Ohm's law using the scalar electric potential as the dependent variable. For dielectric layers the interface solves Gauss' law (it effectively turns into a local Electrostatics interface).

The Electric Currents in Shells interface is very similar to The Electric Currents in Layered Shells Interface. The main difference is the default mode. The Electric Currents in Layered Shells Interface is set to be *layered* by default, while the Electric Currents in Shells interface is set to be *nonlayered* by default.

This interface supports layered shell features, which means each boundary in the 3D geometry may represent a number of thin layers with different electrical properties. Potential variations in the normal direction (in between and inside layers) are supported. For a general description of layer and interface selections, see Layer and Interface Selection Tools. For more information, see the section on how to set up a Layered Material node in the *COMSOL Multiphysics Reference Manual*.

For a more extensive introduction to the physics and equations implemented by this physics interface, see The Electric Currents Interface, and the Theory for the Electric Currents Interface. Additional information is available in Theory for the Electrostatics Interface chapter and The Electric Currents, Single Layer Shell Interface chapter.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `ecis`.

BOUNDARY SELECTION

By default, all boundaries are available for the application of The Electric Currents in Shells Interface. Select the **Restrict to layered boundaries** checkbox to make the interface applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, `slmat1`), otherwise the boundary is marked as not applicable.

SHELL PROPERTIES

The default setting of the combo box **Shell type** is the main difference between The Electric Currents in Shells Interface and The Electric Currents in Layered Shells Interface, where the former is set to be **Nonlayered shell** by default and the later is set to be **Layered shell** by default.

When the **Nonlayered shell** option is selected, the interface treats the thickness as a multiplicative factor which enters the conservation of surface currents.

- If the **Restrict to layered boundaries** checkbox is not selected in the **Boundary Selection** section, a nonlayered material may be defined on the selected boundaries, and the **Thickness** L_{th} can be set as a user defined value or expression. This value overrides the values set in the material nodes.

- You can visualize the selected layered materials and layers in each layered material by clicking the **Layer cross section preview** and **Layer 3D preview** buttons.

When the **Layered shell** option is selected, the Extra Dimension tool is used to solve the equations through the thickness of a layered material. Depending on other settings, it can describe the physics of objects where electric potential vary both tangentially and perpendicularly (through the thickness).

The thickness of the layered material should be set as follows, depending on the type of material:

- In a **Material** node, the **Layer thickness** can be in the table found under the **Material Contents** section of the material **Settings** window. This automatically adds a **Shell** subnode under the **Material** node, transforming it as a layered material.

- When the layered material is a **Single Layer Material**, the **Thickness** is set in the **Layer Definition** section of the **Shell Property Group** window.

- For a general **Layered Material**, added through a **Layered Material Link** or a **Layered Material Stack**, the **Thickness** is set in the **Layer Definition** section of the **Settings** window. Several layers may be defined in the table, and the **Thickness** should be defined for each of them. The total thickness of the layered material is the sum of all the layers thicknesses.

Note that the **Layered shell** option should be used whenever a layered material is applied on the boundaries, because the thickness is part of the material settings.

Clear the **Use all layers** checkbox to apply The Electric Currents in Shells Interface on some layers only. Select a **Layered material** from the list (the interface is then applicable only on the boundaries where this latter material is defined), and clear the checkboxes corresponding to layers where the interface should not be applied in the **Selection** table.

Ⓠ	• The layer thickness variable, `ecis.ds`, used in the weak equations, is the product of the thickness set in the **Shell Properties** section, `ecis.lth`, and the scale factor `ecis.lsc`, which is equal to 1 by default, and can be overridden by a user defined value in a single layer material. • The layer thickness variable, `ecis.ds`, is defined in all dimensions, and is unrelated to the **Out-of-plane thickness** of the layered material, `htlsh.d`, which is only available in 2D.

Ⓠ	• See Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by explicitly or implicitly using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

The desired selection for the physics interface may correspond to boundaries with different layered materials. This means that a single physics interface can accommodate several different stacking sequences.

🔍 See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the *COMSOL Multiphysics Reference Manual* for details on the definition of layered materials.

DISCRETIZATION

Select the shape order for the **Electric potential** dependent variable — **Linear**, **Quadratic** (the default), **Cubic**, **Quartic**, or **Quintic**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

DEPENDENT VARIABLES

The dependent 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.

Layer and Interface Selection Tools

Almost all physics nodes that you add in The Electric Currents in Shells Interface have either a section named **Shell Properties**, or **Layer Selection** or **Interface Selection**. Selections there interact with the standard selections of geometrical objects (boundaries and edges) in order to provide a complete specification of where a material property or boundary condition is to be applied within a layered material.

The physics features can be of two, fundamentally different, types: those who act on the layers themselves, and those who act on the interfaces between layers. The settings for these two types of features are slightly different.

LAYER SELECTIONS

When a physics feature acts on layers, the interpretations of layer selections are:

• A boundary selection in the geometry corresponds to a domain contribution in the selected layers.

• An edge selection in the geometry corresponds to boundary contributions on the faces of the layers exposed to that edge.

This is illustrated by Figure 4-1 and Figure 4-2, where the patch at the lower left is the actual shell, and the 3D sketch shows an expanded view of what it would represent in the physical world.
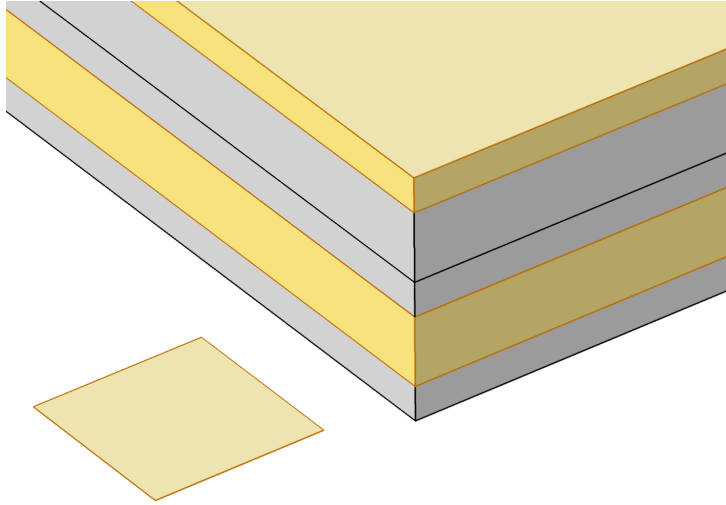
Figure 4-1: Boundary selection with layers 2 and 5 selected.
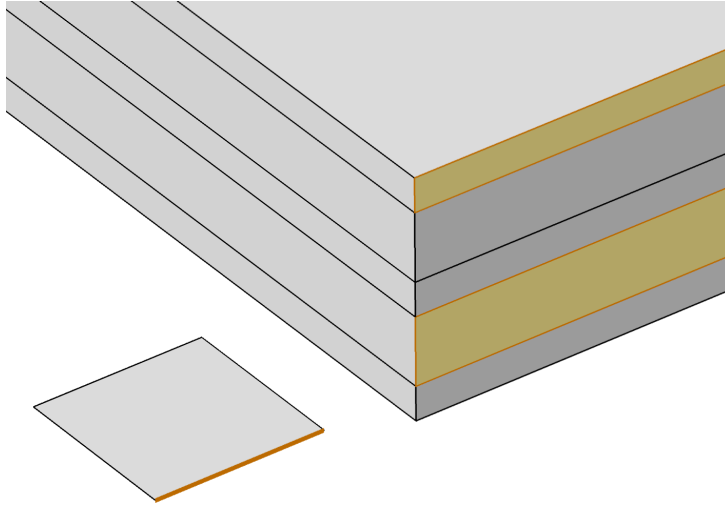
Figure 4-2: Edge selection with layers 2 and 5 selected.

In the **Shell Properties** section, it is possible to limit the contributions to individually selected layers of the selections made in the **Boundary Selection** section. For a given **Layered Material Link**, **Layered Material Stack**, or **Single Layer Material**, you get access to a list of checkboxes for the selection of the individual layers.

INTERFACE SELECTIONS

When a physics feature acts on interfaces, the interpretation of interface selections are:

- A boundary selection in the geometry corresponds to boundary contributions on the selected interfaces.
- An edge selection in the geometry corresponds to edge contributions on the interface edges parallel to the selected edge.

This is illustrated by Figure 4-3 and Figure 4-4, where the patch at the lower left is the actual shell, and the 3D sketch shows an expanded view of what it would represent in the physical world.
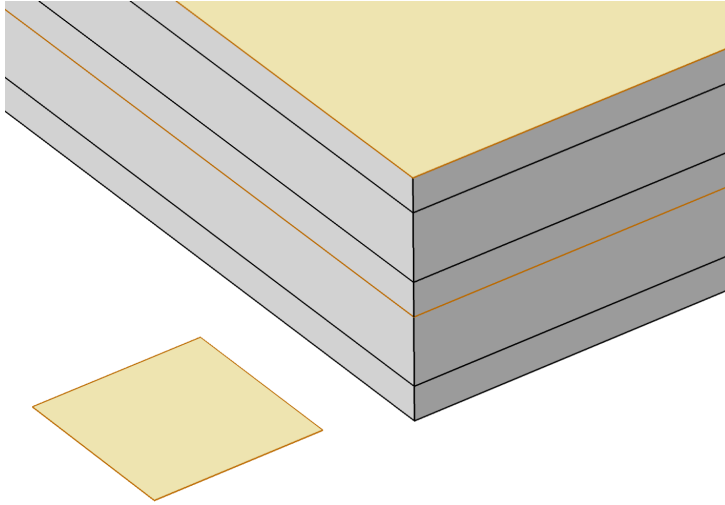
Figure 4-3: Boundary selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.
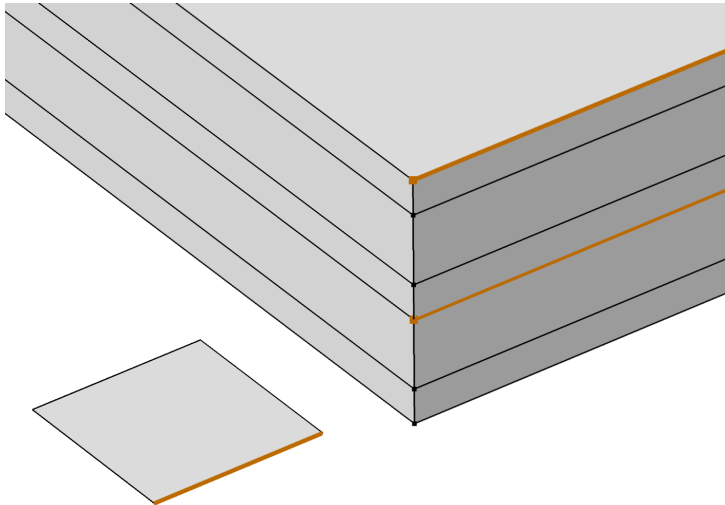
Figure 4-4: Edge selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.

In the **Interface Selection** section, it is possible to select the interfaces between layers where the contribution should be added, including the top and bottom interfaces (see Figure 4-5 for the interfaces description). For a given **Layered Material Link**, **Layered Material Stack**, or **Single Layer Material**, you get access to a list of predefined interfaces subsets.
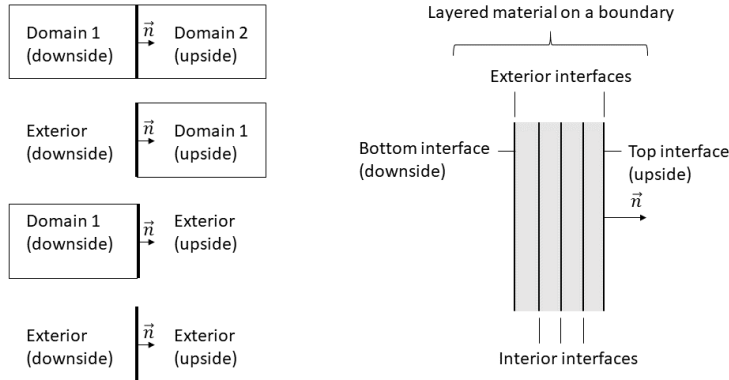
Figure 4-5: Definition of upside and downside settings for a boundary for different configurations, and corresponding interfaces for a layered material.

🔍 See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the *COMSOL Multiphysics Reference Manual*.

Using the Extra Dimension Coordinates

It is possible to write expressions that are functions of the coordinates in the thickness direction of the shell. Expressions based on the usual coordinates, like x, y, and z, are evaluated on the reference surface (the meshed boundaries). In addition to this, the locations in the through-thickness direction can be accessed by making explicit or implicit use of the coordinates in the extra dimension.

The extra dimension coordinate has a name like x_llmat1_xdim. The middle part of the coordinate name is derived from the tag of the layered material definition where it is created; in this example a **Layered Material Link**.

Boundary and Edge Nodes for the Electric Currents in Shells Interface

The Electric Currents in Shells Interface has these boundary and edge nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

> In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

ABOUT THE EDGE CONDITION SELECTIONS

The edge conditions available in The Electric Currents in Shells Interface are applicable to exterior edges only. For shell interfaces, *exterior* means exterior to the boundary selection. In other words; an edge that is connected to adjacent boundaries, only on one side. Consequently, an edge can be regarded as interior (and therefore, not applicable), even though it is *exterior* to the domain that is enclosed by the shell boundary selection. See Layer and Interface Selection Tools for more information.

AVAILABLE NODES

These nodes and subnodes are described in this section:

- Initial Values
- Conductive Shell
- Piezoresistive Shell
- Insulating Layer
- Boundary Current Source
- Interface Current Source

- Interlayer Connection
- Dielectric Layer
- Piezoelectric Layer
- Continuity (Layered Shell)

These nodes are available for this physics interface and described for the Electric Currents and Electrostatics interfaces (listed in alphabetical order):

- Current Source
- Electric Insulation
- Electric Potential
- External Current Density

- Ground
- Normal Current Density
- Terminal
- Floating Potential

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.

INITIAL VALUES

Enter a value or expression for the initial value of the **Electric potential** V (SI unit: V). The default is 0 V.

Conductive Shell

The **Conductive Shell** node is the default feature The Electric Currents in Shells Interface. Its property depends on if the **Shell type** in the physics interface is set to **Nonlayered shell** or **Layered shell**. See the description of **Shell Properties** at the beginning of The Electric Currents in Shells Interface chapter for more information. Generally, the node adds the continuity equation for the electric potential and allows for defining the electric conductivity of the layers. This feature closely relates to Current Conservation under The Electric Currents Interface.

LAYER MODEL

When the **Shell Type** is set to **Layered shell**, by default the **Layer Model** is set to **General** which considers a full equivalent 3D representation of the model. For this case, the **Layer Model** section provides an extra option **Highly conductive layer**. By selecting the **Highly conductive layer**, the feature enforces the potential to be constant through the thickness of the selected layers. For conducting layers where very small electric potential gradients exist in the perpendicular direction, such a setting could increase numerical stability.

OUT-OF-PLANE MATERIAL ORIENTATION

This section is present only when the **Shell Type** is set to **Layered shell**. For the material data inputs, you can select to **Use the laminate coordinate system with** the following three options:

- **No change** (default)
- **Swapped normal and 1st tangential direction**
- **Swapped normal and 2st tangential direction**

When the **Shell Type** on the physics interface is set to **Nonlayered shell**, this section will be replaced by a standard **Coordinate System Selection** section.

Piezoresistive Shell

The **Piezoresistive Shell** feature is an extension of the Conductive Shell. It offers the possibility to set up a multiphysics coupling with the corresponding structural **Shell** and **Layered Shell** physics interfaces to model piezoresistive devices, for which the electric conductivity depends on the mechanical stress or strain.

This feature requires the MEMS module. More information can be found in The Piezoresistivity, Shell Interface and The Piezoresistivity, Layered Shell Interface sections of the *MEMS Module User's Guide*.

LAYER MODEL

When the **Shell Type** is set to **Layered shell**, by default the **Layer Model** is set to **General** which considers a full equivalent 3D representation of the model. For this case, the **Layer Model** section provides an extra option **Highly conductive layer**. By selecting the **Highly conductive layer**, the feature enforces the potential to be constant through the thickness of the selected layers. For conducting layers where very small electric potential gradients exist in the perpendicular direction, such a setting could increase numerical stability.

When working with layered shells, you almost invariably take the material data from what has been defined using **Layered Material Link**, **Layered Material Stack**, or **Single Layer Material** nodes. It is however possible to override some data from a **Piezoresistive Shell** node too. In order to have a correct model, all layers must have been assigned material data for all boundaries selected in the settings for the interface. You can have several **Piezoresistive Shell** nodes with the same (or partially overlapping) geometrical selections, but with different layer selections.

OUT-OF-PLANE MATERIAL ORIENTATION

This section is present only when the **Shell Type** is set to **Layered shell**. For the material data inputs, you can select to **Use the laminate coordinate system with** the following three options:

- **No change** (default)
- **Swapped normal and 1st tangential direction**
- **Swapped normal and 2st tangential direction**

When the **Shell Type** on the physics interface is set to **Nonlayered shell**, this section will be replaced by a standard **Coordinate System Selection** section.

Select a **Constitutive model** — **Piezoresistance form** or **Elastoresistance form**. For each of the following, the default uses values **From material**. For **User defined** enter other values in the matrix or field.

- Specify an **Electric conductivity, zero stress** (SI unit: S/m). This typically comes from the material added under the Materials node.

- For **Piezoresistance form**, select a **Piezoresistance coupling matrix** Π_l (SI unit: $\Omega{\cdot}\text{m}/$ Pa).

- For a **Elastoresistance form**, select an **Elastoresistance coupling matrix** M_l (SI unit: $\Omega{\cdot}\text{m}$).

> ⚠️ When the **Piezoresistive Shell** node is added in the absence of an active multiphysics **Piezoresistivity** coupling node, the material will behave similarly to a Conductive Shell node with the electric conductivity corresponding to the entered value of electric conductivity at zero stress. The piezoresistive effect will be *not* included in the corresponding equation system.

Insulating Layer

The **Insulating Layer** offers the option to not have any current through the selected layer, which is equivalent to removing the physics from the selected layer from the 3D point of view. See Layer and Interface Selection Tools for more information about how to select a layer.

Boundary Current Source

The **Boundary Current Source** node adds a current source Q_j on the boundary.

$$\mathbf{t} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = Q_{j,\,s}$$

It is applicable to interior boundaries that represent either a source or a sink of current.

BOUNDARY CURRENT SOURCE

Enter a value or expression for the **Boundary current source** $Q_{j,s}$ (SI unit: A/m^2). The default is 0 A/m^2.

Interface Current Source

The **Interface Current Source** is available for boundaries only. Instead of representing a source interior to the layer, or a source on the layer's outer perimeter, it represents a net current density flowing from the adjacent domain into the shell ($Q_{j,s}$, SI unit: A/m^2), It can be used to model, for example, a boundary acting as an electrode. Alternatively, for interior interface selections it can represent a source of current on an interface between layers.

In the 3D representation of the layer stack, you could consider it a Boundary Current Source applied to either the top or bottom of the layer stack or on an interface between layers. For more information on the 3D representation of the layer stack, see Layer and Interface Selection Tools.

⚠	Use this node with caution as it might violate the current conservation law that is inherent in Maxwell–Ampère's law.

INTERFACE SELECTION

Select the interfaces for which the source contribution should be added.

🔍	For a general description of layer and interface selections, see Layer and Interface Selection Tools.

INTERFACE CURRENT SOURCE

Enter a value or expression for the **Current source** $Q_{j,s}$ (SI unit: A/m^2). The default is 0 A/m^2.

Interlayer Connection

The **Interlayer Connection** is available only on edges, where it removes for the selected layers, the potential gradient in the perpendicular direction (in the direction of the *extra dimension*). The layers will become electrically connected, and the top and bottom of all involved layers will have the same potential. Multiple **Interlayer Connection** features can be applied to create groups of connected layers.

An **Interlayer Connection** on a single layer can be used to locally convert the full 3D formulation into an in-plane conservation problem. For thin highly conductive layers, this is numerically more stable.

For a general description of layer and interface selections, see Layer and Interface Selection Tools.

CONSTRAINT SETTINGS
To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

Dielectric Layer

The **Dielectric Layer** feature is only available for **Layered Shell**. The node adds the Poisson's equation for the electric potential and allows for defining the electric permittivity of the layers. This feature closely relates to Charge Conservation under The Electrostatics Interface.

Piezoelectric Layer

The **Piezoelectric Layer** node is an extension of the Dielectric Layer to model piezoelectric devices. It is normally used together with a **Piezoelectricity, Layered** multiphysics coupling node and a corresponding **Piezoelectric Material** node in the structural **Layered Shell** interface. This node is added by default to the **Electric Currents in Layered Shells** interface when adding a **Piezoelectricity, Layered Shell** multiphysics interface.

This feature requires one of these products: Structural Mechanics Module, MEMS Module, or Acoustics Module.

Continuity (Layered Shell)

The **Continuity** feature is only available for the interface where the **Layer Type** sets to **Layered Shell**. Use this node to specify the side-by-side continuity conditions on the electric potential at an edge between two layered materials. By default, the electric potential is discontinuous at the interface between two layered materials.

LAYER SELECTION
Select **Source** and **Destination** layered materials from the lists. This makes the node applicable on the edges adjacent to both selected layered materials. Click the **Layer Cross Section Preview** button to visualize the settings.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

CONNECTION SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**. Select either **Straight** (default) or **Twisted** as the **Connection type**.

Floating Potential

The **Floating Potential** node is used when modeling a metallic electrode at floating potential. It applies a constant voltage on the boundary, such that the total normal electric current density equals to zero. Although locally the current density may vary, the total current entering or leaving the boundary equals zero. The constant boundary voltage implies that the tangential electric field equals zero too. Thus, the boundary will behave as an unconnected perfect conductor (a floating equipotential). This is a good approximation when the conductivity of the electrode is many orders of magnitude larger than that of the surrounding medium.

The **Initial value for voltage** V_{init} input is used to provide the initial value for the internal state variables in the Floating Potential. This input can be useful to provide an initial guess to the nonlinear solver or the initial value in a time-dependent study, when the initial values of the other dependent variables (such as the electric potential) are computed from a separate physics or study. In most cases, the default zero value is appropriate.

Electric Currents Solid to Shell Via Boundary

The **Electric Currents Solid to Shell Via Boundary** multiphysics coupling assures continuity of the electric potential between The Electric Currents in Shells Interface and The Electric Currents Interface. The **Electric Currents Solid to Shell Via Boundary** node is available from the **Multiphysics** menu in the **Physics** toolbar or by right-clicking the **Multiphysics Couplings** node in **Model Builder**, if both The Electric Currents in Shells Interface and The Electric Currents Interface are available.

The Electric Currents in Layered Shells Interface

The **Electric Currents in Layered Shells (ecis)** interface ( ), found under the **AC/DC >**
Electric Fields and Currents branch when adding a physics interface, is used to compute
electric field, current, and potential distributions in thin and thick conductive shells
and dielectrics, optionally with an internal layered structure. Inductive effects are
assumed to be negligible; that is, the skin depth should be much larger than the studied
device. It supports stationary, time-dependent, and frequency-domain modeling on
faces in 3D.

The Electric Currents in Layered Shells interface is very similar to The Electric
Currents in Shells Interface. The main difference is the default mode. The Electric
Currents in Shells Interface is set to be nonlayered by default, while the Electric
Currents in Layered Shells interface is set to be layered by default.

🔍	For a more extensive introduction to the physics interface, see The Electric Currents in Shells Interface chapter.

The Electric Currents, Single Layer Shell Interface

The **Electric Currents, Single Layer Shell (ecs)** interface ( ), which is a legacy interface used to is a legacy interface used to compute electric field, current, and potential distributions in thin conducting shells under conditions where inductive effects are negligible; that is, when the skin depth is much larger than the studied device. It supports stationary, frequency-domain, small-signal analysis, and time-domain modeling on edges in 2D and faces in 3D

> 📄 The model with this interface can still be opened and solved. The interface cannot be added as a new interface, and it has been replaced by The Electric Currents in Shells Interface, which is found under the **AC/DC** branch ( ).

The physics interface solves a current conservation equation based on Ohm's law using the scalar electric potential as the dependent variable.

Current Conservation is the main node, which adds the equation for the electric potential and provides a Settings window for defining the electric conductivity as well as the constitutive relation for the electric displacement field and its associated material properties such as the relative permittivity.

When this physics 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**. Then, from the **Physics** toolbar, add other nodes that implement, for example, edge or point conditions and current sources. You can also

right-click **Electric Currents, Single Layer Shell** to select physics features from the context menu.

Except where described here, the majority of the **Settings** windows are the same as for the Electrostatics and Electric Currents interfaces as referenced. The only difference are:

- For 3D components, boundaries are selected instead of domains, and edges instead of boundaries.
- For 2D and 2D axisymmetric components, boundaries are selected instead of domains, and points instead of edges and boundaries.

For a more extensive introduction to the physics and equations implemented by this physics interface, see the Theory for the Electric Currents, Single Layer Shell Interface.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is ecs.

OUT-OF-PLANE THICKNESS

For 2D components, 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.

<table>
<tr>
<td>📝</td>
<td>

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 physics 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)**.

</td>
</tr>
</table>

MANUAL TERMINAL SWEEP SETTINGS

Enter a **Reference impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω.

Select the **Activate manual terminal sweep** checkbox 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 a **Parameter format (value pairs)** for the Touchstone export — **Magnitude and angle (MA)** (the default), **Magnitude in dB and angle (DB)**, or **Real and imaginary parts (RI)**. Select an option from the **If file exists** list — **Overwrite** or **Create new**. Select a **Parameter to export** — **Z** (the default), **Y**, or **S**.

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.

⊙	• Computing Lumped Parameters • Boundary, Edge, Point, and Pair Nodes for the Electric Currents, Single Layer Shell Interface

Boundary, Edge, Point, and Pair Nodes for the Electric Currents, Single Layer Shell Interface

The Electric Currents, Single Layer Shell Interface has these boundary, edge, point, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

💡	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

ABOUT THE EDGE AND POINT CONDITIONS

The nodes in Table 4-1 are available at interfaces between different media and interior edges (in 3D components) and point conditions (in 2D and 2D axisymmetric components) in continuity; that is,

$$\mathbf{n}_2 \cdot (\mathbf{J}_1 - \mathbf{J}_2) = 0$$

which is the natural edge/point condition.

AVAILABLE NODES

These nodes and subnodes are described in this section:

- Change Shell Thickness
- Current Conservation
- Current Source
- Electric Shielding
- Initial Values
- Normal Current Density

These nodes are available for this physics interface and described for the Electric Currents and Electrostatics interfaces (listed in alphabetical order):

- Archie's Law
- Boundary Current Source
- Change Thickness (Out-of-Plane)
- Current Source
- Distributed Impedance
- Effective Medium
- Electric Insulation
- Electric Point Dipole
- Electric Potential
- External Current Density
- Ground
- Initial Values
- Line Current Source
- Point Current Source
- Terminal

Current Conservation

The **Current Conservation** node adds the continuity equation for the electric potential and provides an interface for defining the electric conductivity as well as the constitutive relation and the relative permittivity for the displacement current.

MATERIAL TYPE

The **Material type** setting decides how materials behave and how material properties are interpreted when the mesh is deformed. Select **Solid** for materials whose properties change as functions of material strain, material orientation and other variables evaluated in a material reference configuration (material frame). Select **Nonsolid** for materials whose properties are defined only as functions of the current local state at each point in the spatial frame, and for which no unique material reference configuration can be defined. Select **From material** to pick up the corresponding setting from the domain material on each domain.

ELECTRIC FIELD

See **Electric Field** as described for the Charge Conservation node for the Electrostatics interface.

CONDUCTION CURRENT

By default, the **Electric conductivity** σ (SI unit: S/m) for the media is defined **From material**. Or select **User defined**, **Linearized resistivity**, **Effective medium**, or **Archie's law**.

> ⚠️ Note that general anisotropy is not supported. The boundary orientation has to be normal to one of the principal directions of the material tensor, that is parallel to a plane spanned by the two other principal directions. Violating this condition will result in incorrect results without any warning or error message.

User Defined

For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** depending on the characteristics of the electric conductivity, and then enter values or expressions in the field or matrix.

> 📝 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:

$$\sigma = \frac{1}{\rho_0(1 + \alpha(T - T_{\text{ref}}))}$$

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: $\Omega \cdot$m) are taken **From material**, which means that the values are taken from the domain material.

> 📝 Only certain material models support the **Linearized resistivity**. See AC/DC Material Library in the *COMSOL Multiphysics Reference Manual*.

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.

When **Effective medium** is selected, the Effective Medium subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. When **Archie's law** is selected, an Archie's Law subnode is available.

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. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter a value or expression for the initial value of the **Electric potential** V (SI unit: V). The default is 0 V.

Current Source

The **Current Source** node adds a distributed current source Q_j in the equation that the physics interface defines. Use this node with caution as it might violate the current conservation law that is inherent in Maxwell-Ampère's law.

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or click **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

ELECTRODE CURRENT SOURCE

Enter a value or expression for the **Current source** Q_j (SI unit: A/m^3). The default is $0 \, A/m^3$.

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, Single Layer Shell** interface is defined.

CHANGE SHELL THICKNESS

Enter a **Shell thickness** d_s (SI unit: m). The default is 1 cm.

Normal Current Density

The **Normal Current Density** node represents a current density flowing normally from the domain into the shell boundary. This condition is different from the Current Source node 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.

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or click **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

NORMAL CURRENT DENSITY

Enter a value or expression for the **Normal current density** J_n (SI unit: A/m^2). The default is 0 A/m^2.

Electric Shielding

The **Electric Shielding** node, available for 3D components, can be used to model 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 electric conductivity of σ and a relative permittivity of ε_r.

ELECTRIC SHIELDING

The default **Relative permittivity** ε_r (dimensionless) and **Electric conductivity** σ (SI unit: S/m) take values **From material**. For **User defined** enter different values or expressions.

Enter a value or expression for the **Layer thickness** d_l (SI unit: m). The default is 1 cm.

The Electric Currents, Layered Shell Interface

The **Electric Currents, Layered Shell (ecls)** interface (![icon]), which is a legacy interface used to compute electric fields, currents, and potential distributions in thin conducting layered shells under conditions where inductive effects are negligible; that is, when the skin depth is much larger than the studied device. It supports stationary modeling on faces in 3D.

> The model with this interface can still be opened and solved. The interface cannot be added as a new interface and it has been replaced by The Electric Currents in Layered Shells Interface, which is found under the **Electric Fields and Currents** folder under the **AC/DC** branch (![icon]).

The physics interface solves a current conservation equation based on Ohm's law using the scalar electric potential as the dependent variable. The electric potential is added either by the Layered Shell feature or the Homogenized Shell feature. Both features have advantages and disadvantages, particularly considering shell thickness and electrical connections between the shells.

For (moderately) thick shells, the *layered shell* implementation is superior as it provides a full 3D representation (perpendicular potential gradients and -currents are included by means of an *extra dimension*). The *homogenized shell* implementation instead only solves for in-plane potential gradients. It requires less degrees of freedom, and is numerically more stable. It is especially useful for very thin layers, for which the layered shell implementation may fail. Moreover, it naturally preserves electrical continuity across edges interior to the shell. Both the **Layered Shell** feature and the **Homogenized Shell** feature are equipped with a number of subfeatures, allowing you to take advantage of the strength of each, in different parts of the model.

When the physics interface is added, the following default nodes are added to the **Model Builder** — **Layered Shell** and **Electric Insulation** (the default edge condition). Then, from the **Physics** toolbar, other nodes can be added that implement, for example, boundary

or edge conditions and interlayer connections. You can also right-click **Electric Currents, Layered Shell** to select physics features from the context menu.

This interface supports the Layered Shell feature, which means each boundary may represent a number of thin layers with different electrical properties. Potential variations in the normal direction (in between and inside layers) are supported. For a general description of layer and interface selections, see The Layer Selection Section. See also the section on how to set up a Layered Material node in the *COMSOL Multiphysics Reference Manual*.

For a more extensive introduction to the physics and equations implemented by this physics interface, see The Electric Currents Interface, and the Theory for the Electric Currents Interface. Additional information is available in The Electric Currents, Single Layer Shell Interface chapter.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is ecls.

Select the applicable layers (the default setting is **All layered materials**). If no layered materials have been included yet, there is a shorthand available for creating a **Single Layer Material** (the plus, next to the **Layer Selection** setting).

🔍 • For a general description of layer and interface selections, see The Layer Selection Section.

• You can provide material parameters with a through-thickness variation by explicitly or implicitly using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

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. This variable is explicitly used only by **Homogenized Shell** features. The **Layered Shell** features have their own variable name (for the degrees of freedom in the extra dimension).

DISCRETIZATION

The setting for the discretization order is shared by the **Homogenized Shell** degrees of freedom and the **Layered Shell** degrees of freedom.

The Layer Selection Section

Almost all physics nodes that you add in the Electric Currents, Layered Shell interface have a section named **Layer Selection**. Selections there interact with the standard selections of geometrical objects (boundaries and edges) in order to provide a complete specification of where a material property or boundary condition is to be applied.

The physics features can be of two, fundamentally different, types: Those which act on the layers themselves, and those which act on the interfaces between layers. The settings for these two types of features are slightly different.

The default selections in the **Layer Selection** section differ between different physics features. This reflects the fact that some physics features (such as the Layered Shell feature) are more likely to be applied to all layers, whereas other features (such as the

Homogenized Shell Connection) are more likely to be used for a single layer or interface.

In the **Layer** list, you can select

- **All layered materials**. This option is not available for all features.
- Any **Layered Material Link** defined under **Materials** in the current component.
- Any **Layered Material Stack** defined under **Materials** in the current component.
- Any **Single Layer Material** defined under **Materials** in the current component.

If you select **All layered materials**, you cannot control individual layers; the contribution is given to all layers. All information is taken from the definitions made in the layered material features (**Layered Material Link**, **Layered Material Stack**, or **Single Layer Material**) under **Materials** in the current component. This means that a single physics node can accommodate several different stacking sequences. As geometrical selection, you can use any object selected in any of the layered material features.

If you select an individual **Layered Material Link**, **Layered Material Stack**, or **Single Layer Material**, then you can only select geometrical objects which are part of the selections of that feature. In most cases, you will get access to a list of checkboxes where you can further limit the contributions to individually selected layers or interfaces to which this contribution is to be added.

LAYER SELECTIONS

When a physics feature acts on layers, the interpretation of layer selections are:

- A boundary selection in the geometry corresponds to a domain contribution in the selected layers.
- An edge selection in the geometry corresponds to boundary contributions on the faces of the layers exposed to that edge.

This is illustrated by Figure 4-6 and Figure 4-7, where the patch at the lower left is the actual shell, and the 3D sketch shows an expanded view of what it would represent in the physical world.
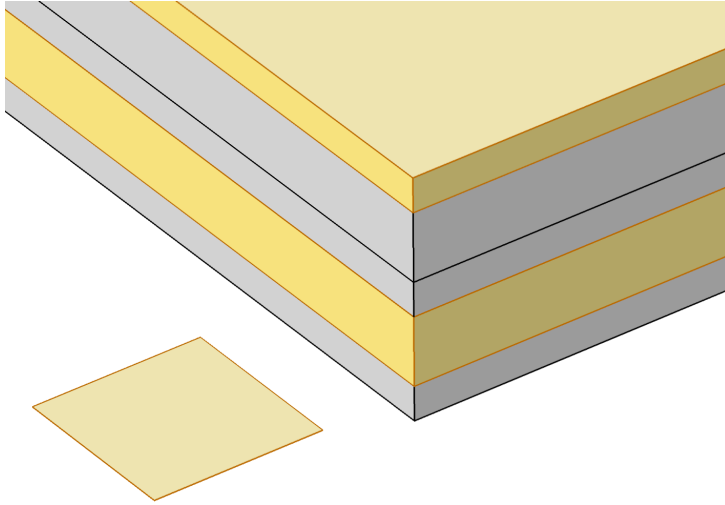
Figure 4-6: Boundary selection with layers 2 and 5 selected.
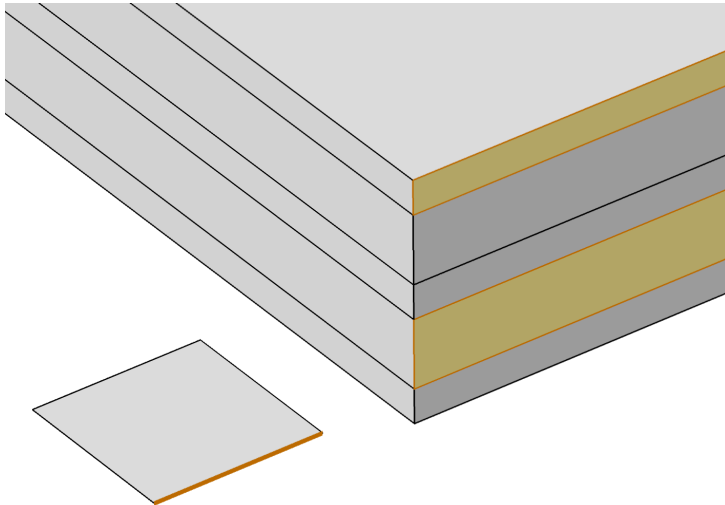
Figure 4-7: Edge selection with layers 2 and 5 selected.

When a physics feature acts on interfaces, the interpretation of interface selections are:

- A boundary selection in the geometry corresponds to boundary contributions on the selected interfaces.

- An edge selection in the geometry corresponds to edge contributions on the interface edges parallel to the selected edge.

This is illustrated by Figure 4-8 and Figure 4-9, where the patch at the lower left is the actual shell, and the 3D sketch shows an expanded view of what it would represent in the physical world.
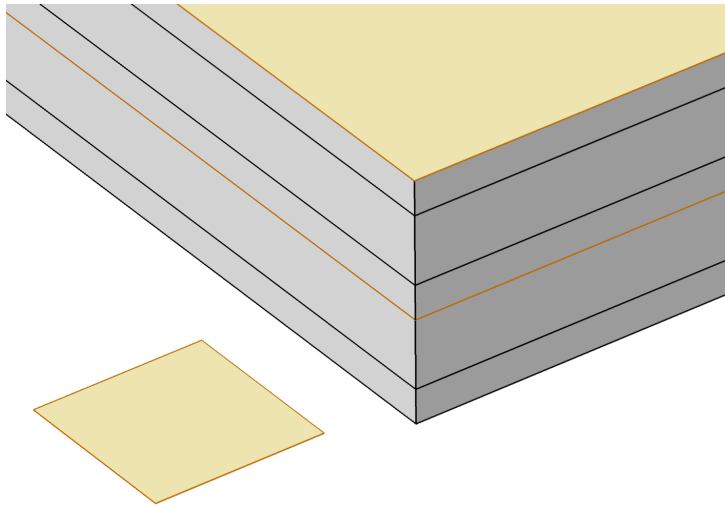
Figure 4-8: Boundary selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.
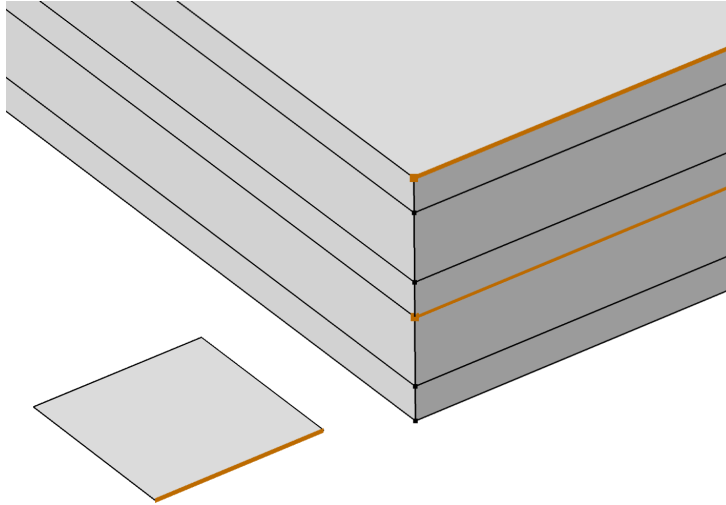
Figure 4-9: Edge selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.

Electrical Continuity in the Electric Currents, Layered Shell Interface

Electric potential in Electric Currents, Layered Shell Interface can be described either by the Homogenized Shell, or Layered Shell feature.

• The **Homogenized Shell** feature imposes electrical continuity among all adjacent homogenized shells, regardless whether they belong to the same **Layered Material**.

• The **Layered Shell** feature exclusively imposes electrical continuity within layered shells sharing the same **Layered Material**.

• The **Layered Shell** feature and the **Homogenized Shell** feature are electrically insulated by default (even when sharing the same **Layered Material**). To override this insulation you can use the Homogenized Shell Connection feature.

For more information; these cases are demonstrated in the *Solid to Multilayered Shell Comparison* model available in the Application Library.

Using the Extra Dimension Coordinates

Sometimes, you want to write expressions that are functions of the coordinates in the thickness direction of the shell. If you write expressions based on the usual coordinates,

like x, y, and z, such an expression will be evaluated on the reference surface (the meshed boundaries). In addition to this, you can access locations in the through-thickness direction by making explicit or implicit use of the coordinates in the extra dimension.

The extra dimension coordinate has a name like x_llmat1_xdim. The middle part of the coordinate name is derived from the tag of the layered material definition where it is created; in this example a **Layered Material Link**.

Boundary and Edge Nodes for the Electric Currents, Layered Shell Interface

The Electric Currents, Layered Shell Interface has these boundary and edge nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

> In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

ABOUT THE EDGE CONDITION SELECTIONS

The edge conditions available in the Electric Currents, Layered Shell Interface are applicable to exterior edges only. For shell interfaces, *exterior* means exterior to the boundary selection. In other words; an edge that is connected to adjacent boundaries, only on one side. Consequently, an edge can be regarded as interior (and therefore, not applicable), even though it is *exterior* to the domain that is enclosed by the shell boundary selection.

AVAILABLE NODES

These nodes and subnodes are described in this section:

- Layered Shell
- Interlayer Connection
- Homogenized Shell Connection
- Homogenized Shell

- Varying Thickness
- Layer Current Source
- Interface Current Source

These nodes are available for this physics interface and described for the Electric Currents and Electrostatics interfaces (listed in alphabetical order):

- Electric Insulation
- Electric Potential
- External Current Density
- Ground
- Terminal

Layered Shell

The **Layered Shell** node adds the continuity equation for the electric potential and allows for defining the electric conductivity of the layers. This interface closely relates to **Current Conservation** under The Electric Currents Interface. Even though the interface works on the boundary level, the resulting equations correspond directly to a volumetric model. The **Layered Shell** includes an extra dimension to describe perpendicular gradients.

Notice that the **Layered Shell** requires a **Layered Material**. This has consequences for the continuity equation. For more information about continuity handling, see Electrical Continuity in the Electric Currents, Layered Shell Interface.

LAYER SELECTION

Select the applicable layers (the default setting is **All layered materials**). Additionally, there is a shorthand available for creating a **Single Layer Material** (the plus, next to the **Layer Selection** setting).

🔍	- For a general description of layer and interface selections, see The Layer Selection Section. - The default **Layered Shell** feature will be locked to the physics interface selection. You can add additional **Layered Shell** features for specific layers, overriding the default properties. - You can provide material parameters with a through-thickness variation by explicitly or implicitly using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

CONDUCTION CURRENT

By default, the **Electric conductivity** σ (SI unit: S/m) of the layers is defined as **From material**. If you select **User defined**, the same user defined value will be applied to all selected layers.

> ⚠️ Note that, as opposed to the Current Conservation feature in the Electric Currents, Single Layer Shell Interface, and the Homogenized Shell feature in the Electric Currents, Layered Shell Interface, the Layered Shell feature does support general anisotropy.

User Defined

For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** depending on the characteristics of the electric conductivity, and then enter values or expressions in the field or matrix.

Interlayer Connection

The **Interlayer Connection** is available both for boundaries, and edges. In both cases it removes for the selected layers, the potential gradient in the perpendicular direction (in the direction of the *extra dimension*). The layers will become electrically connected, and the top and bottom of all involved layers will have the same potential. Multiple **Interlayer Connection** features can be applied to create groups of connected layers.

An **Interlayer Connection** on a single layer can be used to locally convert the full 3D formulation into an in-plane conservation problem. For thin highly conductive layers, this is numerically more stable.

LAYER SELECTION

Select the applicable layers. The layered material is given by the **Layered Shell** parent feature.

> 🔍 For a general description of layer and interface selections, see The Layer Selection Section.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

Homogenized Shell Connection

The **Homogenized Shell Connection** is available for edges. It can be used to override the natural default electric insulation between **Layered Shell** and **Homogenized Shell** features. For more information about continuity of electric potential in Electric Currents, Layered Shell interface, go to Electrical Continuity in the Electric Currents, Layered Shell Interface.

The **Homogenized Shell Connection** removes for the selected layers, the potential gradient in the direction normal to the shell (in the direction of the *extra dimension*), by setting the potential equal to the one in the *base selection*. The base selection potential is the one used by the Homogenized Shell feature. Setting the potential equal to the base potential creates an electrical connection between the selected layers, and **Homogenized Shell** boundaries adjacent to the selected edge.

LAYER SELECTION

Select the applicable layers. The layered material is given by the **Layered Shell** parent feature.

> For a general description of layer and interface selections, see The Layer Selection Section.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

Homogenized Shell

The **Homogenized Shell** feature takes the material properties from one or more selected layers, and determines their effective material properties. It integrates the conductivity in the direction normal to the shell (in the direction of the *extra dimension*), taking the local layer thickness into account:

$$\sigma_{\text{eff}} = \frac{1}{d} \int_{\text{XD}} \sigma dx \,,$$

where d refers to the total thickness of all layers involved. For a discrete set of layers having their own uniform material properties, this integral effectively translates to a sum of conductivities, multiplied by their corresponding local layer thickness:

$$\sigma_{\text{eff}} = \frac{1}{d} \sum_{i=1}^{n} d_{1,i} \sigma_i .$$

This represents a parallel connection of resistors. The **Varying Thickness** subfeature can be used to locally affect the degree to which a particular layer contributes to the sum.

- If composite, layered materials and general anisotropy are not of interest to you and you want to solve an in-shell current conservation problem (as with The Electric Currents, Single Layer Shell Interface), use a **Homogenized Shell** feature with a **Single Layer Material**.
- The Varying Thickness subfeature will allow you to set a spatially or temporally varying thickness (to model material deposition and such).
- The electric potential is automatically continuous among adjacent **Homogenized Shells**. More information about continuity in Electric Currents, Layered Shell interface can be found in the chapter Electrical Continuity in the Electric Currents, Layered Shell Interface.

The resulting material properties are then used in a current conservation problem, that only considers the potential gradients tangential to the shell (in-plane conservation law):

$$\nabla_{\text{T}} \cdot (d \sigma_{\text{eff}} \mathbf{E}_{\text{T}}) = \nabla_{\text{T}} \cdot \mathbf{J}_s = \int_{\text{XD}} Q_s dx ,$$

where the product $d\sigma_{\text{eff}}\mathbf{E}_{\text{T}}$ is effectively the surface current density $\mathbf{J}_{\text{s}}$ in A/m. $\mathbf{E}_{\text{T}}$ is the tangential electric field, and Q_{s} represents a volumetric source of current. Without sources, the right-hand side becomes zero, and in-plane current should be conserved.

When applied to a single layer, the **Homogenized Shell** is virtually identical to the traditional, nonlayered Current Conservation feature in the Electric Currents, Single Layer Shell Interface.

LAYER SELECTION

Select the layers to be included in the integral for the effective material properties.

For a general description of layer and interface selections, see The Layer Selection Section.

By default, the **Electric conductivity** σ (SI unit: S/m) of the layers is defined as **From material**. If you select **User defined**, the same user defined value will be applied to all selected layers.

Varying Thickness

The **Varying Thickness** feature is a subfeature to the **Homogenized Shell** feature. It allows to set a spatially or temporally varying thickness. This is typically used for modeling material deposition and such. For more information on varying thickness and the homogenized shell, see the Homogenized Shell feature.

LAYER SELECTION

Select the layers for which the thickness should be varied.

🔍	For a general description of layer and interface selections, see The Layer Selection Section.

VARYING THICKNESS

Enter a value or expression for the **Selected layers thickness** d_l (SI unit: m). May depend on location, time, temperature, or other quantities. The default value is 0.1 mm.

Layer Current Source

The **Layer Current Source** is available both for boundaries, and edges. In case of a boundary, it represents a volumetric source of current $Q_{j,v}$ inside the selected layers (SI unit: A/m^3).

In case of an edge, it represents an influx of current $Q_{j,s}$ from the edge into the neighboring layers (SI unit: A/m^2). In the 3D representation of the layer stack, you could consider it a "boundary current source" applied to one of the boundaries on the outer perimeter of the layer. For more information on the 3D representation of the layer stack, see The Layer Selection Section.

⚠️	Use this node with caution as it might violate the current conservation law that is inherent in Maxwell–Ampère's law.

LAYER SELECTION

Select the layers for which the source contribution should be added.

	For a general description of layer and interface selections, see The Layer Selection Section.

LAYER CURRENT SOURCE

Enter a value or expression for the **Current source** $Q_{j,v}$ or $Q_{j,s}$ (SI unit: A/m^3 or A/m^2, respectively). For the boundary feature, the default is 0 A/m^3, for an edge feature, the default is 0 A/m^2.

Interface Current Source

The **Interface Current Source** is available for boundaries only. It is similar to the Layer Current Source, but instead of representing a source interior to the layer, or a source on the layer's outer perimeter, it represents a net current density flowing from the adjacent domain into the shell ($Q_{j,s}$, SI unit: A/m^2), It can be used to model, for example, a boundary acting as an electrode. Alternatively, for interior interface selections it can represent a source of current on an interface between layers.

In the 3D representation of the layer stack, you could consider it a "boundary current source" applied to either the top or bottom of the layer stack or on an interface between layers. For more information on the 3D representation of the layer stack, see The Layer Selection Section.

⚠	Use this node with caution as it might violate the current conservation law that is inherent in Maxwell–Ampère's law.

INTERFACE SELECTION

Select the interfaces for which the source contribution should be added.

	For a general description of layer and interface selections, see The Layer Selection Section.

Enter a value or expression for the **Current source** $Q_{j,s}$ (SI unit: A/m^2). The default is 0 A/m^2.

Electric Currents Solid to Shell Via Boundary

The **Electric Currents Solid to Shell Via Boundary** multiphysics coupling assures continuity of the electric potential between The Electric Currents, Layered Shell Interface and The Electric Currents Interface. The **Electric Currents Solid to Shell Via Boundary** node is available from the **Multiphysics** menu in the **Physics** toolbar or by right-clicking the **Multiphysics Couplings** node in **Model Builder**, if both The Electric Currents, Layered Shell Interface and The Electric Currents Interface are available.

5

Magnetic Field Interfaces

This chapter summarizes the functionality of the magnetic field interfaces found under the **Magnetic Fields, No Currents** and **Electromagnetic Fields** folders under the **AC/DC** branch (⚡). It also includes one interface under the **Electromagnetics and Mechanics** folder.

In this chapter:

- The Magnetic Fields, No Currents Interface
- The Magnetic Fields, No Currents, Boundary Elements Interface
- The Magnetic Fields Interface
- The Magnetic and Electric Fields Interface
- The Magnetic Field Formulation Interface
- The Magnetic Fields, Currents Only Interface
- The Rotating Machinery, Magnetic Interface
- The Magnetic Machinery, Rotating, Time Periodic Interface

The Magnetic Fields, No Currents Interface

The **Magnetic Fields, No Currents (mfnc)** interface (⬛), found under the **AC/DC >
Magnetic Fields, No Currents** branch when adding a physics interface, is used to compute
magnetostatic fields from permanent magnets and other current free magnetic sources.
The formulation is stationary but for use together with other physics also time-domain
modeling is supported in 2D and 3D.

The physics interface solves Gauss' law for the magnetic field using the scalar magnetic
potential as the dependent variable.

The main node 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 and all the magnetic fields are originated by permanent magnets
or external systems not included in the model.

When this physics interface is added, these default nodes are also added to the **Model
Builder** — **Free Space**, **Magnetic Insulation** (the default boundary condition), and **Initial
Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example,
additional boundary conditions and point conditions. You can also right-click **Magnetic
Fields, No Currents** to select physics features from the context menu.

Physics-Controlled Mesh

The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the
Sequence type is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled
Mesh** section, find the physics interface in the **Contributor** column and select or clear
the checkbox in the **Use** column on the same table row for enabling (the default) or
disabling contributions from the physics interface to the physics-controlled mesh.

Information from the physics, such as the presence of an infinite elements domain or
periodic condition, will be used to automatically set up an appropriate meshing
sequence.

> ⓘ In the *COMSOL Multiphysics Reference Manual* see the
> Physics-Controlled Mesh section for more information about how to
> define the physics-controlled mesh.

	Except where described below, the **Settings** windows are described for the Magnetic Fields and Electrostatics interfaces.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is mfnc.

BACKGROUND MAGNETIC FIELD

Select an option from the **Solve for** list — **Reduced field** or **Full field** (the default). For **Reduced field** specify a **Background magnetic field** $\mathbf{H}_b$ (SI unit: A/m). The entered expressions must be differentiable.

	Notice that expressions including coupling operators are not differentiable and cannot be used as background fields.

When a background field is active, the physics interface solves for the reduced (perturbation) field only. This setting, together with the External Magnetic Flux Density feature, is useful to introduce an external field generated by systems not included in the model.

THICKNESS

	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 components.

ERROR CHECK

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

When the **Check applicability of features in study** checkbox is selected, any features that are incompatible with the study will generate an error message when trying to solve or show the default solver. No solver will be generated. Deselect it and you will be able to run the model, possibly with runtime errors instead. It is available to allow the advanced user to tweak any feature and use it outside of its intended study scope.

DEPENDENT VARIABLES

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

DISCRETIZATION

Select the shape order for the **Magnetic scalar potential** dependent variable — **Linear**, **Quadratic** (the default), or **Cubic**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

🔍	• Domain, Boundary, Point, and Pair Nodes for the Magnetic Fields, No Currents Interface • Theory for the Magnetic Fields, No Currents Interface

▥	• *Magnetic Prospecting of Iron Ore Deposits*: Application Library path **ACDC_Module/Introductory_Magnetostatics/magnetic_prospecting** • *Magnetic Signature of a Submarine*: Application Library path **ACDC_Module/Introductory_Magnetostatics/magnetic_signature_submarine**

Domain, Boundary, Point, and Pair Nodes for the Magnetic Fields, No Currents Interface

The Magnetic Fields, Currents Only Interface has these domain, boundary, point, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

	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 physics 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 NODES

These nodes are available for this physics interface and listed in alphabetical order. Also see Table 5-1 for a list of interior and exterior boundary conditions.

- External Magnetic Flux Density
- Initial Values
- Free Space
- Force Calculation[1]
- Laminated Core
- Magnet
- Magnetic Flux Conservation
- Magnetic Flux Density
- Magnetic Insulation

- Magnetic Scalar Potential
- Magnetic Shielding
- Periodic Condition
- Sector Symmetry[1]
- Symmetry Plane (for Magnetic Field)
- Thin Low Permeability Gap
- Zero Magnetic Scalar Potential

[1] Described for another physics interface in this user guide.

	- Infinite Elements, Perfectly Matched Layers, and Absorbing Layers in the *COMSOL Multiphysics Reference Manual*

Table 5-1 lists the interior and exterior boundary conditions available with this physics interface.

TABLE 5-1: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS FOR THE MAGNETIC FIELDS, NO CURRENTS INTERFACE.

PHYSICS NODE	INTERIOR	EXTERIOR
External Magnetic Flux Density		x
Magnetic Flux Density		x
Magnetic Insulation	x	x
Magnetic Scalar Potential	x	x
Magnetic Shielding	x	x
Periodic Condition		x
Symmetry Plane		x
Thin Low Permeability Gap	x	
Zero Magnetic Scalar Potential	x	x

Free Space

For electromagnetic modeling, it is quite common for the fields to extend into the environment of the device in question. It is therefore important not only to model the device itself, but also its surroundings. Typically, this is done by adding a box or sphere around the device's geometry.

The default **Free Space** node is used to specify the physical conditions in close proximity to the device — typically in air or vacuum. It adds an **Magnetic Flux Conservation** governing equation using the **Relative permeability** constitutive relation with the permeability μr set to one.

The **Free Space** feature provides a canvas on top of which other features can be added to locally specify material properties and excitation forms.

Magnetic Flux Conservation

The **Magnetic Flux Conservation** node adds Equation 2-22 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). There are two types of **Magnetic Flux Conservation** available; **Magnetic Flux Conservation in Solids** and **Magnetic Flux Conservation in Fluids**. This distinction decides how materials behave and how material properties are interpreted when the mesh is deformed.

Magnetic Flux Conservation in Solids applies to materials whose properties change as functions of material strain, material orientation, and other variables evaluated in a material reference configuration (material frame).

Magnetic Flux Conservation in Fluids applies to materials whose properties are defined only as functions of the current local state at each point in the spatial frame, and for which no unique material reference configuration can be defined.

!	In COMSOL versions **6.2** and earlier, this **Solids/Fluids** distinction was controlled within the **Magnetic Flux Conservation** node by specifying **Solid/Nonsolid** in the **Material type** setting.

See the settings for Constitutive Relation B-H under Ampère's Law, except for these differences:

- **Magnetic losses** and **Effective B-H curve**, being time-harmonic material models, are not available.

- Select **B-H curve** to use a curve that relates magnetic flux density **B** and the magnetic field **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** node adds an initial value for the magnetic scalar potential that can serve as an initial guess for a nonlinear solver.

INITIAL VALUES

Enter a value or expression for the initial value of the **Magnetic scalar potential** V_m. The default value is 0 A.

Magnetic Insulation

The **Magnetic Insulation** node 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.

Magnetic Scalar Potential

The **Magnetic Scalar Potential** node adds a boundary condition for the magnetic scalar potential:

$$V_m = V_{m0}$$

Enter a value or expression for the **Magnetic scalar potential** V_{m0} (SI unit: A). The default is 0 A.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

Magnet

The **Magnet** node is used to model one or multiple magnets, simplifying the definition of the magnetization direction in each magnet. In The Rotating Machinery, Magnetic Interface, this feature is called **Nonconducting Magnet**.

MAGNET

Direction Method
Select the **Direction method** to be used for determining the direction of the magnetization in the magnet. Choose between:

- **Specify north and south boundaries**, where the north and south boundaries of the magnet can be selected. The direction of the magnetization is then calculated based on the position average of each pole.

- **Specify by domains**, which allows the magnetization direction to be set individually in different subselections of the **Magnet** feature. The magnetization in each such domain can then be set by either specifying the north and south boundaries, or by entering a user defined direction.

- **User defined**, which allows the user to manually specify a direction vector for the magnetization.

Pattern Type
This section is only available when **Specify north and south boundaries** is selected as the **Direction method**. The setting makes it possible to automatically set the magnetization direction for multiple magnets following a certain pattern. Choose between **No pattern**, **Circular pattern**, and **Linear pattern (or domain index based)**. When **Circular pattern** is selected, the magnetization direction for each pattern element is determined by its position in a circle and a pattern repetition angle. Select **Linear pattern (or domain index based)** to repeat the definition of the magnetization direction in a selection of magnets that are linearly displaced.

Type of Periodicity
This section is only available when **Circular pattern** or **Linear pattern (or domain index based)** is chosen as the **Pattern type**. The different options are **Repeating**, **Alternating**, and **User defined**. When **Repeating** is selected, the magnetization direction is the same in each element. Select **Alternating** to instead have the magnetization direction reversed for each repeated element in the pattern. If the alternating pattern is applied to an odd number of magnets, the north and south boundaries should be specified for the first magnet in the pattern. The **User defined** option is only available for a circular pattern and allows a custom input that determines how much the magnetization direction should change between each element.

Pattern Additional Angle
This section is only available when **User defined** is chosen as the **Type of periodicity**, when also having **Circular pattern** as the **Pattern type**. This angle is added to the magnetization direction for each repetition in the pattern.

Pattern Center
This section is only available when **Circular pattern** is chosen as the **Pattern type**. It defines the center of the circular pattern.

Pattern Axis
This section is only available when **Circular pattern** is chosen as the **Pattern type**. This setting is only available in 3D, and defines the axis around which the pattern is created.

Direction of Magnetization
This section is only available when **User defined** is selected as the **Direction method** in the **Magnet** feature. This input vector is equivalent to the one that can be added when **Magnetization** is selected in the Magnetic Flux Conservation feature.

CONSTITUTIVE RELATION B-H
This section is identical to the one in the Magnetic Flux Conservation node, apart from that only the options **Remanent flux density** and **Nonlinear permanent magnet** are available here.

Laminated Core

The **Laminated Core** node is used to model an anisotropic effective material for laminated soft magnetic cores that is not developing induced currents. The **Magnetization model** can be set to **B-H Curve** (default), **Effective B-H Curve**, and **Relative Permeability**, which all have the same settings as described in the Ampère's Law node

of the Magnetic Fields interface. The **Lamination model** setting is used to choose between **Homogeneous** (default), **Out-of-plane** (available in 2D only), and **Direction based** types of lamination. The first is used to describe uniform material cores, as described in the Magnetic Flux Conservation node in the Magnetic Fields, No Currents interface. The latter are used to reproduce the homogenized effect of a stack of soft iron laminates and insulation spacers along a direction of lamination, based upon the Energy- and Coenergy-Based Magnetic Constitutive Relations. The direction of lamination and the stacking factor (the volumetric ratio between the laminates and the total stack) can be set. These two quantities are used to evaluate the effective magnetic energy density of the stack and the corresponding effective magnetic field.

It is also possible to specify the **Loss Model**. When enabled, it allows to define the **Volumetric loss density** (SI unit: W/m^3) and can be specified as **User Defined**, or, if **B-H Curve** is chosen, by using empirical loss formulas via **Steinmetz** or **Bertotti** coefficients. More information on loss models can be found in the Loss Calculation node documentation. In the The Rotating Machinery, Magnetic Interface, this feature is called Laminated Core, Magnetic Flux Conservation.

Magnetic Flux Density

The **Magnetic Flux Density** node 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:

$$-\mathbf{n} \cdot \mathbf{B} = B_n \tag{5-2}$$

Using this formulation, it is possible to specify the normal component of the magnetic flux density as a scalar.

Select a **Type** of the boundary condition to use — **Inward flux density** (the default) or **Magnetic Flux density**:

- For **Inward flux density** 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.

- For **Magnetic flux density** 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** node provides a boundary condition that specifies a zero magnetic potential on the boundary $V_m = 0$.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

Symmetry Plane (for Magnetic Field)

The **Symmetry Plane** node adds a boundary condition that represents symmetry or antisymmetry in the magnetic field, depending on which option is chosen.

SYMMETRY TYPE

Choose between **Symmetry** and **Antisymmetry** for the magnetic field. If **Symmetry** is chosen, the boundary condition is

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

which states that the normal component of the magnetic field is zero. If **Antisymmetry** is chosen, the boundary condition becomes

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

which states that the tangential components of the magnetic field are zero.

▥	*Permanent Magnet*: Application Library path **ACDC_Module/ Introductory_Magnetostatics/permanent_magnet**

External Magnetic Flux Density

📄 This node 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** in the physics 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.

Magnetic Shielding

The **Magnetic Shielding** node adds a boundary condition for magnetic shielding. It describes a thin layer of a highly permeable medium that shields the magnetic field. The boundary condition uses the following equations for the normal magnetic flux density discontinuity and the tangential magnetic field inside the layer:

$$\mathbf{n} \cdot (\mathbf{B}_1 - \mathbf{B}_2) = \nabla_t \cdot (d_s \mathbf{B}_t(\mathbf{H}_t))$$
$$\mathbf{H}_t = -\nabla_t V_m$$

In these equations, ∇_t represents a tangential derivative (gradient), and d_s is the surface thickness.

MAGNETIC SHIELDING

Enter a value or expression for the **Surface thickness** d_s (SI unit: m).

Relative Permeability
Select **Relative permeability** μ_r (dimensionless) to use the constitutive relation $\mathbf{B} = \mu_0 \mu_r \mathbf{H}$. The default uses values **From material**. For **User defined** enter another value or expression.

B-H Curve
Select **B-H curve** $|\mathbf{B}|$ (SI unit: T) to use a curve that relates magnetic field $\mathbf{H}$ and the magnetic flux density $\mathbf{B}$ as $|\mathbf{B}| = f(|\mathbf{H}|)$. The **Magnetic flux density norm** setting can take the values **From material** or **User defined**.

Magnetic Scalar Potential Discontinuity

The **Magnetic Scalar Potential Discontinuity** node adds a boundary condition on an interior boundary which allows for a prescribed discontinuity in the magnetic scalar potential. The difference in magnetic scalar potential through this boundary is given by

$$V_m^u - V_m^d = V_{m,d}$$

where $V_m{}^u$ is the magnetic scalar potential on the upside of the boundary selection and $V_m{}^d$ is the magnetic scalar potential on the downside.

When applied to a cut boundary making a nonsimply connected region simply connected, this feature formally represents a surface current flowing along the boundaries connecting the upside and downside boundaries.

Thin Low Permeability Gap

The **Thin Low Permeability Gap** node adds a boundary condition on an interior boundary which allows for a discontinuity in the magnetic scalar potential. This feature represents a thin layer in a magnetic material filled with another material that has smaller relative magnetic permeability. Use this feature to model, for example, an air gap in a magnetic circuit. Enter a relative magnetic permeability μ_r for the thin layer material as well as a thickness d_s. The difference in magnetic flux through this boundary is given by

$$\mathbf{n} \cdot \mathbf{B}_1 = \mu_0 \mu_r \left(\frac{V_{m,1} - V_{m,2}}{d_s} \right) = -\mathbf{n} \cdot \mathbf{B}_2$$

where $V_{m,1}$ is the magnetic scalar potential on the upside of the boundary selection and $V_{m,2}$ is the magnetic scalar potential on the downside.

THIN LOW PERMEABILITY GAP

The default **Relative permeability** μ_r (dimensionless) uses values **From material**. For **User defined** enter a different value or expression. Enter a value or expression for the **Surface thickness** d_s (SI unit: m).

Archie's Law

See the Archie's Law section in the Electric Field and Current Interfaces chapter.

The Magnetic Fields, No Currents, Boundary Elements Interface

The **Magnetic Fields, No Currents, Boundary Elements (mfncbe)** interface ( ), found under the **AC/DC** > **Magnetic Fields, No Currents** branch when adding a physics interface, is used to compute magnetostatic fields from permanent magnets and other current free magnetic sources. It can also be coupled to The Magnetic Fields, Currents Only Interface or The Magnetic Fields Interface to account for outer, current free regions as an alternative to using Infinite Elements. The formulation is stationary. The physics interface solves Gauss' Law for the magnetic field using the scalar magnetic potential as the dependent variable.

$$-\nabla \cdot (\mu_0 \mu_r \nabla V_m) = 0 .$$

	For a discussion about the boundary element method, see Theory for the Boundary Elements PDE in the *COMSOL Multiphysics Reference Manual*.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Magnetic Flux Conservation**, **Magnetic Insulation** (on exterior boundaries), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, magnetic scalar potential and magnetic flux density conditions. You can also right-click **Magnetic Fields, No Currents, Boundary Elements** to select physics features from the context menu.

	If both The Magnetic Fields, Currents Only Interface and The Magnetic Fields, No Currents, Boundary Elements Interface are available, the Magnetic Scalar-Scalar Potential Coupling node is available from the **Multiphysics** menu in the **Physics** toolbar or by right-clicking the **Multiphysics Couplings** node in **Model Builder**.

If both The Magnetic Fields Interface and The Magnetic Fields, No Currents, Boundary Elements Interface are available, the Magnetic Vector-Scalar Potential Coupling node is available from the **Multiphysics** menu in the **Physics** toolbar or by right-clicking the **Multiphysics Couplings** node in **Model Builder**.

Physics-Controlled Mesh

The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the **Sequence type** is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the checkbox in the **Use** column on the same table row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

Information from the physics will be used to automatically set up an appropriate meshing sequence.

In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh.

DOMAIN SELECTION

From the **Selection** list, select any of the options — **Manual**, **All domains**, **All voids**, or **All domains and voids** (the default). The geometric entity list displays the selected domain entity numbers. Edit the list of selected domain entity numbers using the selection toolbar buttons to the right of the list or by selecting the geometric entities in the **Graphics** window. Entity numbers for voids can be entered by clicking the Paste ( ) button in the selection toolbar and supplying the entity numbers in the dialog. The entity number for the infinite void is 0, and finite voids have negative entity numbers.

Selections can also be entered using the **Selection List** window, available from the **Windows** menu in the **Home** toolbar.

For more information about making selections, see Working with Geometric Entities in the *COMSOL Multiphysics Reference Manual*.

Select the **Enable physics symbols** checkbox to display symmetry planes (in 3D) and lines (in 2D) in the **Graphics** window, as specified in the Symmetry settings.

SYMMETRY

🔷	For 3D components, from the **Condition for the x = x₀ plane**, **Condition for the y = y₀ plane**, and **Condition for the z = z₀ plane** lists, choose **Off** (the default), **Symmetric** (infinite magnetic insulation plane), or **Antisymmetric** (infinite ground plane). Then enter the value for the plane location x_0, y_0, or z_0 (SI unit: m) as required.
🔵	For 2D components, from the **Condition for the x = x₀ plane** and **Condition for the y = y₀ plane** lists, choose **Off** (the default), **Symmetric** (infinite magnetic insulation line), or **Antisymmetric** (infinite ground line). Then enter the value for the plane location x_0 or y_0 (SI unit: m) as required.

FAR FIELD APPROXIMATION

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

📄	For more information about the **Far Field Approximation** settings, see Far-Field Approximation Settings in the *COMSOL Multiphysics Reference Manual*.

QUADRATURE

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

📄	For more information about the **Quadrature** settings, see Quadrature in the *COMSOL Multiphysics Reference Manual*.

THICKNESS (2D COMPONENTS)

For 2D components, enter a default value 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 components.

	For 3D components, select the **Infinity condition** — **Magnetic insulation** (the default) or **Zero magnetic scalar potential at infinity**. If there is an antisymmetric symmetry in the potential field, it acts as an added infinite ground plane with a fixed value of the magnetic scalar potential. The value at infinity is fixed to 0 by the presence of the infinite ground plane, so for this case there is only a fixed **Zero magnetic scalar potential at infinity** condition.
	For 2D components, select the **Infinity condition** — **Magnetic insulation** (the default) or **Zero magnetic scalar potential at reference distance**. If there is an antisymmetric symmetry in the potential field, it acts as an added ground line with a fixed value of the magnetic scalar potential. The value at a reference distance is fixed to 0 by the presence of the ground line, so for this case there is only a fixed **Zero magnetic scalar potential at reference distance** condition.

DISCRETIZATION

From the **Magnetic scalar potential/Magnetic flux density** list, choose from predefined options for the boundary element discretization order for the magnetic scalar potential variable and the magnetic flux density variable, respectively. The predefined options represent the suitable combinations of element orders such as **Quadratic/Linear** (the default).

The settings under **Value types when using splitting of complex variables** are important for sensitivity and optimization computations. See the description of the built-in operators fsens and fsensimag.

DEPENDENT VARIABLES

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

Domain and Boundary Nodes for the Magnetic Fields, No Currents, Boundary Elements Interface

The Magnetic Fields, No Currents, Boundary Elements Interface has these domain and boundary nodes available from the **Physics** ribbon (Windows users), **Physics**

context menu (macOS or Linux users), or by right-clicking the physics interface in the **Model Builder** to access the context menu (all users).

	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

AVAILABLE NODES

AVAILABLE NODES

These nodes and subnodes are described in this section:

- Magnetic Flux Conservation
- Force Calculation[1]
- Magnetic Insulation
- Initial Values
- Zero Magnetic Scalar Potential

- Magnetic Scalar Potential
- Magnetic Flux Density
- Magnetic Scalar-Scalar Potential Coupling
- Magnetic Vector-Scalar Potential Coupling

[1] Described for another physics interface in this user guide.

Magnetic Flux Conservation

The **Magnetic Flux Conservation** node adds Equation 2-22 (or a similar equation depending on the selected constitutive relation) for the magnetic scalar potential and provides an interface for defining the constitutive relation and the relevant material properties (for example, the relative permeability).

CONSTITUTIVE RELATION B-H

	See the settings for Constitutive Relation B-H under Ampère's Law. Notice that the Magnetic Fields, No Currents, Boundary Elements interface supports the following constitutive relations only: **Relative permeability**, **Remanent flux density**, and **Magnetization**.
!	Notice that the boundary element method is based on the availability of an analytic Green's function for the domain. Thus, the **Relative permeability** must evaluate to a constant value for each **Magnetic Flux Conservation** node.

Magnetic Insulation

The **Magnetic Insulation** node for the Magnetic Fields, No Currents, Boundary Elements interface provides the default boundary condition. The magnetic insulation boundary condition sets the normal component of the magnetic flux density to zero:

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

	This condition is useful at boundaries confining a surrounding region of air or to model symmetry cuts.

Initial Values

Initial Values adds an initial value for the magnetic scalar potential that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver. Add more **Initial Values** nodes from the **Physics** toolbar, **Global** menu.

INITIAL VALUES

Enter a value or expression for the initial value of the **Magnetic scalar potential** V_m. The default value is 0 A.

Zero Magnetic Scalar Potential

The **Zero Magnetic Scalar Potential** node provides a boundary condition that specifies a zero magnetic potential on the boundary $V_m = 0$.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

Magnetic Scalar Potential

The **Magnetic Scalar Potential** node adds a boundary condition for the magnetic scalar potential:

$$V_m = V_{m0}$$

Enter a value or expression for the **Magnetic scalar potential** V_{m0} (SI unit: A). The default is 0 A.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

Magnetic Flux Density

The **Magnetic Flux Density** node adds a boundary condition for the magnetic flux density.

MAGNETIC FLUX DENSITY

> 📝 See the settings for Magnetic Flux Density in the Magnetic Fields, No Currents documentation. Notice that the Magnetic Fields, No Currents, Boundary Elements interface supports the **Inward flux density** option only.

Magnetic Scalar-Scalar Potential Coupling

The **Magnetic Scalar-Scalar Potential Coupling** multiphysics node assures continuity of the magnetic scalar potential across boundaries between The Magnetic Fields, Currents Only Interface and The Magnetic Fields, No Currents, Boundary Elements Interface. The **Magnetic Scalar-Scalar Potential Coupling** node is available from the **Multiphysics** menu in the **Physics** toolbar or by right-clicking the **Multiphysics Couplings** node in **Model Builder**, if both The Magnetic Fields, Currents Only Interface and The Magnetic Fields, No Currents, Boundary Elements Interface are available.

> ⚠ The Magnetic Fields, Currents Only Interface can be set to solve for the **Reduced field** using the Background Magnetic Field setting. The Magnetic Fields, No Currents, Boundary Elements Interface does not have such a setting but it can be made to represent an open boundary for the reduced field by adding, in The Magnetic Fields, Currents Only Interface, an External Magnetic Flux Density feature on the boundary selection for the **Magnetic Scalar-Scalar Potential Coupling**.

⚠ When using The Magnetic Fields, No Currents, Boundary Elements Interface with iterative solvers, **Right Preconditioning** is preferred. Thus, it is advised to check that **Right Preconditioning** is used when combining the **Magnetic Scalar-Scalar Potential Coupling** with The Magnetic Fields, No Currents, Boundary Elements Interface and other physics interfaces. How to define **Right Preconditioning** is described in Iterative in the *COMSOL Multiphysics Reference Manual*.

BOUNDARY SELECTION

Select **Manual** or **All boundaries** from the **Selection** list. Make additional edits to the list of boundary entity numbers using the Selection toolbar buttons. When **All boundaries** is selected from the **Selection** list, the boundaries exterior to the Magnetic Fields, No Currents interface that intersect the exterior boundaries to the Magnetic Fields, No Currents, Boundary Elements interface are available in the boundary entity number list.

⚠ When using the symmetry options at the physics interface level as discussed in Symmetry, there will be no degrees of freedom on the symmetry planes. For such cases, please make sure that the coupling selection does not contain boundaries that intersect with the symmetry planes.

COUPLED INTERFACES

Select **Magnetic Fields, No Currents** as **Primary interface** and **Magnetic Fields, No Currents, Boundary Elements** as **Secondary interface**.

Magnetic Vector-Scalar Potential Coupling

The **Magnetic Vector-Scalar Potential Coupling** multiphysics node assures continuity of the magnetic flux density across boundaries between The Magnetic Fields Interface and The Magnetic Fields, No Currents, Boundary Elements Interface. The **Magnetic Vector-Scalar Potential Coupling** node is available from the **Multiphysics** menu in the **Physics** toolbar or by right-clicking the **Multiphysics Couplings** node in **Model Builder**, if

both The Magnetic Fields Interface and The Magnetic Fields, No Currents, Boundary Elements Interface are available.

⚠ The Magnetic Fields Interface can be set to solve for the **Reduced field** using the Background Magnetic Field setting. The Magnetic Fields, No Currents, Boundary Elements Interface does not have such a setting but it can be made to represent an open boundary for the reduced field by adding, in The Magnetic Fields Interface, a Magnetic Field feature with the background magnetic field on the boundary selection for the **Magnetic Vector-Scalar Potential Coupling**.

⚠ When combining The Magnetic Fields, No Currents, Boundary Elements Interface with The Magnetic Fields Interface, the latter has to use Gauge Fixing for A-Field and a solved using Fully Coupled and a Direct solver, see the *COMSOL Multiphysics Reference Manual* for details on Fully Coupled and Direct.

BOUNDARY SELECTION

Select **Manual** or **All boundaries** from the **Selection** list. Make additional edits to the list of boundary entity numbers using the Selection toolbar buttons. When **All boundaries** is selected from the **Selection** list, the boundaries exterior to the Magnetic Fields interface that intersect the exterior boundaries to the Magnetic Fields, No Currents, Boundary Elements interface are available in the boundary entity number list.

⚠ When using the symmetry options at the physics interface level as discussed in Symmetry, there will be no degrees of freedom on the symmetry planes. For such cases, please make sure that the coupling selection does not contain boundaries that intersect with the symmetry planes.

COUPLED INTERFACES

Select **Magnetic Fields** as **Primary interface** and **Magnetic Fields, No Currents, Boundary Elements** as **Secondary interface**.

The Magnetic Fields Interface

The **Magnetic Fields (mf)** interface (🧲), found under the **AC/DC** > **Electromagnetic Fields** branch when adding a physics interface, is used to compute magnetic field and induced current distributions in and around coils, conductors, and magnets. Depending on the licensed products, stationary, frequency-domain, small-signal analysis, and time-domain modeling are supported in 2D and 3D. Note that the frequency and time-domain formulations become ill-posed when approaching the static limit. You can extend the useful frequency range downward by adding a low conductivity.

The physics interface solves Maxwell's equations, which are formulated using the magnetic vector potential and, optionally for coils, the scalar electric potential as the dependent variables.

The main node is the **Free Space** feature, which solves Ampère's law in free space. This adds the equation for the magnetic vector potential and assumes that all domains have a uniform relative magnetic permeability and permittivity of one.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Magnetic Fields**, **Free Space**, **Magnetic Insulation** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement boundary conditions and external currents. You can also right-click **Magnetic Fields** to select physics features from the context menu.

Physics-Controlled Mesh

The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the **Sequence type** is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the checkbox in the **Use** column on the same table row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

Information from the physics, such as the presence of an infinite elements domain or periodic condition, will be used to automatically set up an appropriate meshing sequence.

> [!] In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `mf`.

BACKGROUND FIELD

This section allows the specification of a background magnetic vector potential (that generates a background magnetic flux density).

Select an option to **Solve for** — **Full field** (the default) or **Reduced field**. When **Reduced field** is selected, choose the Background field specification — **Magnetic vector potential** (the default) or **Uniform magnetic flux density**. Mathematically, the background field is always specified as a background magnetic vector potential. The **Uniform magnetic flux density** option will apply an appropriate magnetic vector potential generating the specified magnetic flux density. In axisymmetric geometries, the uniform background field is applied in the axial direction.

For **Magnetic vector potential**, enter values or expressions for the **Background magnetic vector potential** $\mathbf{A}_b$ (SI unit: Wb/m). The defaults are 0 Wb/m. For **Uniform magnetic flux density**, in 2D or 3D components, enter the values or expressions of the components of the **Uniform magnetic flux density** $\mathbf{B}_b$ (SI unit: T); in 2D axisymmetric components, enter the value or expression of the **Uniform axial magnetic flux density** $B_{b,z}$ (SI unit: T). The defaults are 0 T. The specified background magnetic flux densities must be uniform (space-independent), but they can be a function of time.

When a background field is active, this solves for the relative (perturbated) field only. This setting, together with the External Magnetic Vector Potential feature, is useful to introduce an external field generated by systems not included in the model.

COMPONENTS

This section is only available in 2D and 2D axially symmetric components.

The current vector has the same direction as the magnetic vector potential. 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 components.

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.

When **In-plane vector potential** or **Three-component vector potential** is selected, the Gauge Fixing for A-Field node becomes available.

In 2D axisymmetric dimension, The Covariant Formulation is used.

THICKNESS

For 2D components, enter a value or expression for the global **Out-of-plane thickness** d (SI unit: m). The default 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 components.

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.

PORT SWEEP SETTINGS

Select the **Activate port sweep** checkbox to switch on the sweep and invoke a parametric sweep over the Lumped Port features. Enter a **Sweep parameter name** to indicate the name of the model parameter that controls the port activated in each step of the sweep. The default is PortName. The name given must match the model parameter, defined under **Global Definitions**, that is object of a **Parametric Sweep** node in the current **Study**.

The lumped parameters computed can be subject to a **Touchstone file export**. To activate this functionality, enter a file path or click **Browse** to navigate to a file. Select a **Parameter format (value pairs)** for the Touchstone export — **Magnitude and angle (MA)** (the default), **Magnitude in dB and angle (DB)**, or **Real and imaginary parts (RI)**. Select the desired operation to perform from the **If file exists** list — **Overwrite** (the default) or **Create new**. The latter is useful when the model is solved multiple times with different settings. Enter a **Reference impedance, Touchstone file export** Z_{ref} (SI unit: Ω). The default is 50 Ω.

ERROR CHECK

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

When the **Check applicability of features in study** checkbox is selected, any features that are incompatible with the study will generate an error message when trying to solve or show the default solver. No solver will be generated. Clear the checkbox and you will be able to run the model, possibly with runtime errors instead. It is available to allow the advanced user to tweak any feature and use it outside of its intended study scope.

DEPENDENT VARIABLES

The dependent variable is the **Magnetic vector potential** A. You can change both its field name and the individual component variable names. If the new field name coincides with the name of another magnetic vector potential field in the model, the physics interfaces share degrees of freedom and component names. The new field name must not coincide with the name of a field of another type, or with a component name belonging to some other field. Component names must be unique within a model, except for fields of the same type sharing a common field name.

DISCRETIZATION

Select the shape order for the **Magnetic vector potential** dependent variable — **Linear**, **Quadratic** (the default), or **Cubic**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

🔍	• Domain, Boundary, Point, and Pair Nodes for the Magnetic Fields Interface • Computing Lumped Parameters • Theory for the Magnetic Fields Interface

▦	*Quadrupole Lens*: Application Library path **COMSOL_Multiphysics/ Electromagnetics/quadrupole**

▦	*Eddy Currents*: Application Library path **ACDC_Module/ Introductory_Electromagnetics/eddy_currents**

Domain, Boundary, Point, and Pair Nodes for the Magnetic Fields Interface

The Magnetic Fields interface has these domain, boundary, point, and pair nodes available, which are listed in alphabetical order.

About the Boundary Conditions

With no surface currents present, the physics 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 **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.

Table 5-2 lists the interior and exterior boundary conditions available with this physics interface.

TABLE 5-2: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS FOR THE MAGNETIC FIELDS INTERFACE.

NODE	INTERIOR	EXTERIOR
External Magnetic Vector Potential	x	x
Impedance Boundary Condition		x
Layered Transition Boundary Condition	x	
Lumped Port	x	x
Magnetic Field		x
Magnetic Insulation	x	x
Magnetic Potential	x	x
Magnetic Shielding	x	
Perfect Magnetic Conductor	x	x
Periodic Condition		x
Surface Current Density	x	x
Surface Magnetic Current Density	x	x
Symmetry Plane		x
Thin Low Permeability Gap	x	
Transition Boundary Condition	x	

Available Nodes

These nodes, listed in alphabetical order, are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or by right-clicking to access the context menu (all users).

💡	In general, to add a node, go to the **Physics** toolbar no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

Also see Table 5-2 for a list of interior and exterior boundary conditions.

- Free Space
- Ampère's Law
- Ampère's Law, Piezomagnetic
- Ampère's Law, Nonlinear Magnetostrictive
- Boundary Feed
- Coil
- Coil Geometry
- Edge Current
- Electric Point Dipole
- External Current Density
- External Magnetic Vector Potential
- Force Calculation (described for the Electrostatics interface)
- Gap Feed
- Harmonic Perturbation
- Impedance Boundary Condition
- Initial Values
- Layered Transition Boundary Condition
- Line Current (Out-of-Plane)
- Laminated Core
- Lumped Port
- Magnet
- Magnetic Field
- Magnetic Insulation (the default boundary condition)
- Magnetic Point Dipole
- Magnetic Point Dipole (on Axis)
- Magnetic Potential
- Magnetic Shielding
- Passive Conductor
- Perfect Magnetic Conductor
- Periodic Pair
- Sector Symmetry
- Single Conductor Coil
- Surface Current Density
- Surface Magnetic Current Density
- Symmetry Plane (for Magnetic Flux Density)
- Thin Low Permeability Gap
- Transition Boundary Condition
- Velocity (Lorentz Term)

For 2D axisymmetric components, the COMSOL software takes the axial symmetry boundaries (at $r = 0$) into account and adds an **Axial Symmetry** node to the component that is valid on the axial symmetry boundaries only.

Infinite Elements, Perfectly Matched Layers, and Absorbing Layers in the *COMSOL Multiphysics Reference Manual*

!	In the *COMSOL Multiphysics Reference Manual*, see Table 2-4 for links to common sections and Table 2-5 for common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Free Space

For inductive and capacitive (sometimes resistive) analysis, it is quite common for the fields to extend into the environment of the device in question. It is therefore important not only to model the device itself, but also its (close) surroundings. Typically, this is done by adding a box or sphere around the device's geometry.

The default **Free Space** node is used to specify the physical conditions in close proximity to the device — *typically in air or vacuum*. It adds an Ampère's Law *governing equation* with a limited set of material models: the permeability μ_r and permittivity ε_r are assumed to be one, and the conductivity is taken from the **Stabilization conductivity**.

The Free Space feature provides a canvas on top of which the **Ampère's Law** feature and other features can be added to locally specify material properties and excitation forms.

STABILIZATION

For transient and frequency domain modeling of inductive devices, it is desirable to limit the contrast in conductivity. This improves the numerical stability. In reality, the conductivity may vary between 10^{-14} S/m (dry air) and $6 \cdot 10^7$ S/m (for copper), but from a numerical viewpoint any material that causes insignificant *eddy current* or *leakage current* loss is effectively an insulator. As a result, the "insulator conductivity" is often chosen to be much higher than the true material property (note that this reasoning does not hold for capacitive modeling).

A suitable rule of thumb for choosing the stabilization conductivity is given by the skin depth, as compared to the overall geometry size. The assumption is that if the skin depth in **Free Space** is around one hundred times the device size, the resulting loss and the impact on lumped device properties are insignificant. At the same time, the model will still be sufficiently stable. What is considered a good value will differ per model, though. It is therefore recommended to double-check the *free space loss* as compared to other loss terms in the model. If the stabilization conductivity has a negative impact on the overall model accuracy, consider lowering it. If the solver has trouble converging, consider raising it. Alternatively, consider using **Gauge Fixing**.

Stabilization Conductivity

The **Stabilization conductivity** allows for five options:

- **From material**; use the conductivity specified by the material.

- **Automatic** (default in 3D); set the skin depth δ_s to be 100 times the geometry size and use that to deduce the appropriate conductivity σ_{stab}. The **Typical frequency** used for this conversion is taken from the solver. If there is no such frequency available — as is the case for a *transient analysis* — a **Fallback frequency** is used. The appropriate fallback frequency may differ per model and is best determined manually. Note that in addition to the fallback frequency, a solver-independent (user defined) frequency is an option too. A good value is the typical operating frequency of the modeled device. For more information, see Stabilization.

- **From skin depth**; very similar to the option **Automatic**, but with the added advantage that a *user defined* skin depth can be set.

- **Off** (default in 2D); disable the stabilization conductivity altogether: $\sigma_{stab} = 0$. This models a perfect vacuum and is a good approximation for air. In 2D oftentimes only the out-of-plane magnetic vector potential is used, providing sufficient stability.

- **User defined**; specify a suitable stabilization conductivity directly.

⚠️ Note that a **Fallback frequency** that is chosen too low may result in an overly damped model, just like a stabilization conductivity that is chosen too high, or a skin depth that is chosen too small. *The device's typical operating frequency, combined with a skin depth that is at least an order of magnitude larger than the device size, should provide a suitable stabilization conductivity.*

When in doubt, it is recommended to check the amount of resistive loss in the insulators and compare this against the overall device loss. Sweeping over the stabilization conductivity (investigating the effect of different amounts of stabilization) or comparing against a *Gauge Fixed* model with zero stabilization is an option too.

Ampère's Law

The **Ampère's Law** node adds Ampère's law for the magnetic field to a domain and provides an interface for defining the constitutive relation and its associated properties as well as electric properties. There are two types of Ampère's Law available; **Ampère's**

Law in Solids and **Ampère's Law in Fluids**. This distinction decides how materials behave and how material properties are interpreted when the mesh is deformed.

Ampère's Law in Solids applies to materials whose properties change as functions of material strain, material orientation, and other variables evaluated in a material reference configuration (material frame).

Ampère's Law in Fluids applies to materials whose properties are defined only as functions of the current local state at each point in the spatial frame, and for which no unique material reference configuration can be defined.

> | ! | In COMSOL versions **6.1** and earlier, this **Solids/Fluids** distinction was controlled within the **Ampère's Law** node by specifying **Solid/Nonsolid** in the **Material type** setting. |

CONSTITUTIVE RELATION B-H

Specify the constitutive relation that describes 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.

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 a **Magnetization model** — **Relative permeability** (the default), **B-H curve**, **Magnetic losses**, **Remanent flux density**, **Magnetization**, **Effective B-H curve**, **Hysteresis Jiles–Atherton model**, **Nonlinear permanent magnet**, **Analytic magnetization curve**, or **External material**.

> | ! | **Ampère's Law in Fluids** uses only **Relative Permeability** or **Magnetization**. The other constitutive relations are only accessible in **Ampère's Law in Solids**. |

Relative Permeability
Select **Relative permeability** μ_r (dimensionless) to use the constitutive relation $\mathbf{B} = \mu_0\mu_r\mathbf{H}$. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix. If **Effective medium** is selected, an Effective Medium subnode is available from the context menu (right-click the parent node) as well as from the **Physics** toolbar, **Attributes** menu, which can specify the relative permeability of the mixture.

B-H Curve

Select **B-H curve** ‖**H**‖ (SI unit: A/m) to use a curve that relates magnetic flux density **B** and the magnetic field **H** as ‖**H**‖ = f(‖**B**‖). Note that in literature **B** is typically given as a function of **H**, as ‖**B**‖ = f(‖**H**‖)) or, when preserving direction: **B** = f(‖**H**‖)**H**⁄‖**H**‖. For the Ampère's Law feature, however, **B** is closely related to the degree of freedom **A** and then **H** is derived from that.

The **Magnetic field norm** and **Magnetic coenergy density** settings can take the values **From material** or **User defined**.

When **User defined** is selected, specify a user-defined expression for the magnetic field norm. The direction of the magnetic field is taken to be the same as the direction of the magnetic flux density at each point.

!	Do not select this option in a Frequency Domain study, such as when using The Induction Heating Interface. This option is not relevant for time harmonic modeling, which assumes linear material properties.
	Check your own B-H curve data using the *B-H Curve Checker*: Application Library path **ACDC_Module/Applications/bh_curve_checker**

Magnetic Losses

This option introduces a complex relative permeability and it is intended for time-harmonic (frequency domain) studies. Therefore, it is not available for The Magnetic Fields, Currents Only Interface.

Select **Magnetic losses** μ' and μ'' (dimensionless) 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 that the time-harmonic Sign Convention requires a lossy material to have a positive material parameter μ'' (see Modeling Losses in the Frequency Domain).

Remanent Flux Density

Select **Remanent flux density** $\mathbf{B_r}$ (SI unit: T) to use the constitutive relation $\mathbf{B} = \mu_0\mu_{rec}\mathbf{H} + \mathbf{B_r}$, where μ_{rec} and $\mathbf{B_r}$ are the recoil permeability and the remanent flux density respectively (the flux density when no magnetic field is present). The recoil permeability μ_{rec} is very similar to the relative permeability, and is valid as long as the magnet is subjected to normal operating conditions (it is only valid within the linear region close to the vertical axis $\mathbf{H} = 0$). $\mathbf{B_r}$ is given by taking the remanent flux density

norm (typically, provided by the material) and multiplying it with a normalized direction field specified in the physics: $\mathbf{B_r} = \|\mathbf{B_r}\| \, \mathbf{e} / \|\mathbf{e}\|$.

- The default recoil permeability μ_{rec} (dimensionless) uses values **From material**. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the recoil permeability and enter another value or expression in the field or matrix.

- The remanent flux density norm $\|\mathbf{B_r}\|$ is taken **From material** by default. The AC/DC branch in the material library contains a number of hard magnetic materials specifically for this purpose. Alternatively, choose **User defined**, and specify your own expression.

- Enter **x** and **y** components for the **Remanent flux direction e**. For 3D components, enter **x**, **y**, and **z** components.

Magnetization
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 components, enter **x**, **y**, and **z** components.

Effective B-H Curve
Select **Effective B-H curve** $|\mathbf{H}|_{eff}$ (SI unit: A/m) to use an effective curve that provides the local linearized relation between the magnetic flux density $\mathbf{B}$ and the magnetic field $\mathbf{H}$ in time-harmonic problems.

> [!] This option is designed for time-harmonic (frequency-domain) problems and implements an effective saturation model for nonlinear materials. See Effective Nonlinear Magnetic Constitutive Relations for more information.
>
> Generate the effective B-H curve data using the *Effective Nonlinear Magnetic Curves Calculator*: Application Library path **ACDC_Module/ Applications/effective_nonlinear_magnetic_curves**

Nonlinear Permanent Magnet
Select **Nonlinear permanent magnet** to use a nonlinear BH-relation that is isotropic around a point in $\mathbf{H}$ space that is shifted by the coercive field $\mathbf{H_c}$. This constitutive relation is intended for easy modeling of self-demagnetization of soft permanent magnets. It is recommended to keep all input settings at the default **From material** and use the example material, **Nonlinear Permanent Magnet** in the **AC/DC** material library either as is or as a template for defining customized materials. The latter is done by

changing the interpolation functions defined in the material. The example material is a generic and approximate representation of AlNiCo 5.

The **Direction of magnetization** is the only input that normally should be entered in the physics.

!	Do not select this option in a Frequency Domain study, such as when using The Induction Heating Interface. This option is not relevant for time harmonic modeling, which assumes linear material properties.

!	See also Effective Nonlinear Magnetic Constitutive Relations in the theory section.

Hysteresis Jiles–Atherton model
Select the **Hysteresis Jiles–Atherton model** to use in the constitutive relation
$\mathbf{B} = \mu_0\mathbf{H} + \mu_0\mathbf{M}$ with the magnetization $\mathbf{M}$ (SI unit: A/m) computed from the solution of the five parameters Jiles–Atherton model. Specify the five parameters **M$_s$**, **a**, **k**, **c**, and α either from the material (default) or as user defined. The example material **Jiles–Atherton Hysteretic Material** is available in the AC/DC material library. The parameters may be tensor quantities resulting in the modeling of an anisotropic hysteretic material as shown in the application library entry:

▦	*Vector Hysteresis Modeling*: Application Library path **ACDC_Module/Verifications/vector_hysteresis_modeling**

In stationary study, the model defaults to use **Transient initialization** defined by the user. Switch it to **Parametric hysteresis** when performing Parametric Sweep on time instants. The entry **Initial Magnetization** is present to set the initial values of Jiles–Atherton variables.

The **Discretization** section is used to choose the discretization order of Jiles–Atherton variables.

!	For a more through description of the model, its parameters and literature references, see The Jiles–Atherton Hysteresis Model in the theory section.

Analytic Magnetization Curve

Select this option use the constitutive relation $\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M(H)} + \mathbf{M_r})$, where $\mathbf{M_r}$ is the remanent magnetization (SI unit: A/m), and the magnetization vector is $\mathbf{M(H)}$ is calculated from the magnetic field using a nonlinear relation with possible saturation and hysteresis. This option is only available in **Ampère's Law in Solids**.

The material is assumed to exhibit spontaneous magnetization, so that it is constituted of domains with nonzero magnetization even at zero applied field. Application of a magnetic field can rearrange the domains resulting into the net magnetization in the material. At very large applied fields, the magnetization saturates, as all domains in the material become aligned along the direction of the applied field. Domain wall interactions can also lead to a significant hysteresis in the magnetization process.

All domains have magnetization of the same magnitude $|\mathbf{M}| = M_s$, but the magnetization can have different orientations. The applied magnetic field changes the domain orientation, and the resulting net magnetization is found from the following nonlinear implicit relation:

$$\mathbf{M} = M_s L(|\mathbf{H}_{\text{eff}}|)\frac{\mathbf{H}_{\text{eff}}}{|\mathbf{H}_{\text{eff}}|}$$

The effective field in the material is given by

$$\mathbf{H}_{\text{eff}} = \mathbf{H} + \alpha\mathbf{M}$$

where the matrix α characterizes the interdomain coupling.

The magnetization shape is characterized by the function L with the following properties:

For weak effective fields, the magnetization is nearly linear

$$\mathbf{M} \approx \chi_0\mathbf{H}_{\text{eff}}$$

and can be characterized by the initial magnetic susceptibility matrix χ_0.

For strong fields, the magnetization magnitude approaches the saturation value

$$\mathbf{M} \approx M_s\frac{\mathbf{H}_{\text{eff}}}{|\mathbf{H}_{\text{eff}}|}$$

The magnetization shape is characterized be the Langevin function

$$L = \coth\left(\frac{3\chi_0|\mathbf{H}_{\text{eff}}|}{M_s}\right) - \frac{M_s}{3\chi_{0m}|\mathbf{H}_{\text{eff}}|}$$

where χ_0 is the magnetic susceptibility in the initial linear region.

Other possible choices of the L function are a hyperbolic tangent, which is sometimes referred to as the Ising model

$$L = \tanh\left(\frac{\chi_0|\mathbf{H}_{\text{eff}}|}{M_s}\right)$$

and a linear function

$$L = \frac{\chi_0|\mathbf{H}_{\text{eff}}|}{M_s}$$

The latter option will make it possible to find an explicit expression for the magnetization. However, such model does not have a proper saturation behavior, and thus it should be used only in the operating range far from saturation. Both the Langevin function and hyperbolic tangent models requires the magnetization vector components to be treated as extra dependent variables.

Possible hysteresis of in the material magnetization is characterize using the Jiles–Atherton model. The model assumes that the total magnetization can be represented as a sum of hysteretic and anhysteretic parts, the latter one is given by

$$\mathbf{M}_{\text{an}} = M_s L(|\mathbf{H}_{\text{eff}}|)\frac{\mathbf{H}_{\text{eff}}}{|\mathbf{H}_{\text{eff}}|}$$

The change in the total magnetization caused by the change on the effective magnetic field is represented as

$$d\mathbf{M} = c_r d\mathbf{M}_{\text{an}} + \max(\xi \cdot d\mathbf{H}_{\text{eff}}, 0)\xi/|\xi|$$

$$\xi = k_p^{-1}(\mathbf{M}_{\text{an}} - \mathbf{M})$$

where c_r is the reversibility parameter, and k_p is the pining loss parameter. The above can be solved using either a time-dependent analysis or a stationary parametric sweep.

	For more details, see The Jiles–Atherton Hysteresis Model in the theory section.

External Material

Select **External material** to use a curve that relates magnetic flux density **B** and the magnetic field **H** as $|\mathbf{H}| = f(|\mathbf{B}|)$ according to an externally coded function.

Specify the **External material** to use (from the **Materials** node under **Global Definitions**). This setting allows using material models or constitutive relations defined in an external library. See Working with External Materials for more information.

CONSTITUTIVE RELATION JC-E

This section is described for the Current Conservation feature.

The options **Effective medium** and **Archie's law** require additional subnodes. If **Effective medium** is selected, an Effective Medium subnode is available from the context menu (right-click the parent node) as well as from the **Physics** toolbar, **Attributes** menu. If **Archie's law** is selected, add an Archie's Law subnode in the same way. These subnodes contain additional settings to specify how the material properties are computed. **Effective medium** models a mixture of materials whose properties are computed by averaging the properties of the components. **Archie's law** models a conductive liquid in a nonconductive matrix.

CONSTITUTIVE RELATION D-E

The default **Relative permittivity** ε_r (dimensionless) for the media is used **From material** and defined on the shell domain. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the permittivity and then enter values or expressions in the field or matrix. If **Effective medium** is selected, an Effective Medium subnode is available from the context menu (right-click the parent node) as well as from the **Physics** toolbar, **Attributes** menu, which can specify the relative permittivity of the mixture.

Ampère's Law, Magnetostrictive

> | ! |
>
> The **Ampère's Law, Magnetostrictive** node is obsolete and will be removed in future versions. It cannot be added in version 6.1 and later, but may be present in models created by earlier versions of the software.
>
> You are advised to update your model to use one of the new multiphysics interfaces instead, either The Nonlinear Magnetostriction Interface or The Piezomagnetism Interface (for modeling linear magnetostrictive effects).

Ampère's Law, Piezomagnetic

The **Ampère's Law, Piezomagnetic** node adds Ampère's law for the magnetic field in a piezomagnetic material (also called linear magnetostrictive material), and it provides an interface for defining the electric properties. It is normally used as part of the Piezomagnetism multiphysics interface (described in the *Structural Mechanics Module User's Guide*) together with a **Piezomagnetic Effect** multiphysics coupling node and **Piezomagnetic Material** node in the corresponding **Solid Mechanics** interface. **Ampère's Law, Piezomagnetic** node is added by default to the **Magnetic Fields** interface when adding a Piezomagnetic Effect multiphysics interface. The interface requires the AC/DC Module and at least one of the following modules: Structural Mechanics, MEMS, Acoustics.

> ⚠️ When the **Ampère's Law, Piezomagnetic** node is added to the Magnetic Fields interface in the absence of an active **Piezomagnetic Effect** multiphysics coupling node, the material behaves similarly to an **Ampère's Law** node with some limitations. Thus, the magnetic permittivity of free space will be assumed. The piezomagnetic effect is then *not* included in the corresponding equation system.

Ampère's Law, Nonlinear Magnetostrictive

MAGNETIZATION

Specify the constitutive relation that describes the macroscopic properties of the medium (relating the magnetic flux density **B** and the magnetic field **H**) for a magnetostrictive material:

$$\mathbf{B} = \mu_0[\mathbf{H} + \mathbf{M}(\mathbf{H}, S_{\text{mech}}) + \mathbf{M}_r]$$

where $\mathbf{M}(\mathbf{H}, S_{\text{mech}})$ is the material magnetization that depends on the magnetic field and mechanical stress, and $\mathbf{M}_r$ is the remanent magnetization vector (SI unit: A/m).

> 🔍 • Nonlinear Magnetostriction
> • Modeling Magnetostrictive Materials
> • Analytic Magnetization Curve

CONDUCTION CURRENT

This section is described for the Current Conservation feature.

ELECTRIC FIELD

The default **Relative permittivity** ε_r (dimensionless) for the media is used **From material**. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the permittivity and then enter values or expressions in the field or matrix.

Initial Values

The **Initial Values** node 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.

INITIAL VALUES

Enter values or expressions for the **Magnetic vector potential A** (SI unit: Wb/m). The defaults are 0 Wb/m.

External Current Density

The **External Current Density** node 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.

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

EXTERNAL CURRENT DENSITY

Enter a value or expression for each component of the **External current density** $\mathbf{J}_e$ (SI unit: A/m^2). The defaults are 0 A/m^2.

The external current density does not contribute to the losses (due to Joule heating), since there is no electric field associated with it. To include the contribution to the losses from the external current density, select the **Add contribution of the external current density to the losses** checkbox. Then select an option from the **External losses** list — **From domain conductivity** (the default) or **User defined**. If **From domain conductivity** is selected, the heat source is computed using the conductivity specified in

the material model feature (such as **Ampère's Law**) that is applied in the domain. For **User defined**, enter a value for Q_e (SI unit: W/m^3) to specify a user-defined heat source.

Velocity (Lorentz Term)

The **Velocity (Lorentz term)** node adds velocity **v**. The external current is equal to $\sigma\mathbf{v} \times \mathbf{B}$.

This node is available in 2D and 2D axisymmetry space dimensions of the **Magnetic Fields**, **Magnetic and Electric Fields**, and **Rotating Machinery, Magnetic** interfaces. In 2D and 2D axisymmetry, it is available when only solving for the out-of-plane component of the magnetic vector potential. The node is also available in 3D Magnetic and Electric Fields interface when solving for both the electric potential and the magnetic vector potential.

!	Correct use of the velocity feature requires deep physical insight. In situations when the moving domain is of bounded extent in the direction of the motion, material properties vary in this direction, or the domains contains magnetic sources that also move, the Lorentz term must not be used.
	There are several library examples demonstrating the correct usage:
	• *Magnet Falling Through Copper Tube*: Application Library path **ACDC_Module/Devices,_Transducers_and_Actuators/falling_magnet**
	• *Magnetic Brake*: Application Library path **ACDC_Module/ Devices,_Transducers_and_Actuators/magnetic_brake**
	• *Homopolar Generator*: Application Library path **ACDC_Module/ Devices,_Motors_and_Generators/homopolar_generator**

An operational definition of when it can be used is that the moving domain should only contain an induced magnetic source (magnetization plus 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

homogeneous plane (maglev trains); and flow of homogeneous conducting fluid past a magnet (liquid metal pumps or Hall generators/thrusters, for example).

	If you are not sure how to proceed, contact the COMSOL Support Center: www.comsol.com/support.

User defined is selected by default. Enter the components for the **Velocity** vector **v** (SI unit: m/s) or, if present, select any velocity field 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.

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

	• Strictly speaking, the tangential components of the magnetic potential is $(\mathbf{n} \times \mathbf{A}) \times \mathbf{n}$.
!	• Magnetic insulation is a special case of the Magnetic Potential boundary condition that sets the tangential component of the magnetic potential to zero.

This node is used for the modeling of a lossless metallic surface, for example, a ground plane or as a symmetry type boundary condition. The node 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. The node 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 are automatically balanced by induced surface currents.

The Magnetic Insulation node can also be applied on interior boundaries. The boundary will then support two surface current densities on the two sides, denoted bu Jsu (upside) and Jsd (downside).
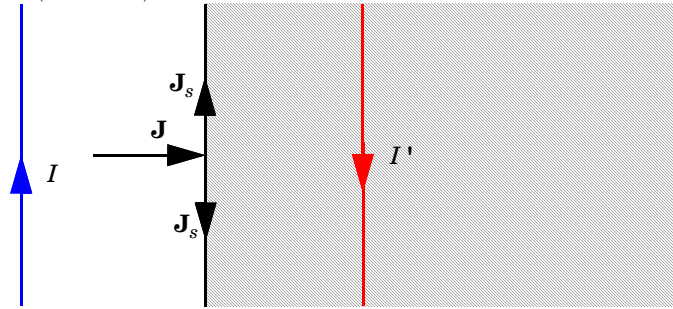
Figure 5-1: The magnetic insulation boundary condition is used on exterior and interior boundaries representing the surface of a lossless metallic conductor or (on exterior boundaries) a symmetry cut. The shaded (metallic) region is not part of the model but still carries effective mirror images of the 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.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

> **!** Weak constraints perform poorly when applied on vector elements. They should be used when the magnetic vector potential is discretized with Lagrange elements, for example, when solving for out-of-plane component in a two-dimensional component.

Magnetic Field

The **Magnetic Field** node adds a boundary condition for specifying the tangential component of the magnetic field at the boundary:

$$\mathbf{n} \times \mathbf{H} = \mathbf{n} \times \mathbf{H}_0$$

The **Harmonic Perturbation** subnode (it is of the exclusive type) is available from the context menu (right-click the parent node) or in the **Physics** toolbar by clicking the **Attributes** menu and selecting **Harmonic Perturbation**. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

MAGNETIC FIELD

Enter the value or expression for the **Magnetic Field** $\mathbf{H}_0$ (SI unit: A/m) vector coordinates. The defaults are 0 A/m.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

Surface Current Density

The **Surface Current** node adds a boundary condition for a surface current density $\mathbf{J}_s$:

$$-\mathbf{n} \times \mathbf{H} = \mathbf{J}_s$$
$$\mathbf{n} \times (\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{J}_s$$

These expressions apply to exterior and interior boundaries respectively. Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual.*

SURFACE CURRENT

Enter values or expressions for the **Surface current density** $\mathbf{J}_{s0}$ (SI unit: A/m) coordinates. The defaults are 0 A/m.

Surface Magnetic Current Density

The **Surface Magnetic Current Density** boundary condition

$$\mathbf{n} \times \mathbf{E} = \mathbf{J}_{ms}$$
$$\mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) = -\mathbf{J}_{ms}$$

specifies a surface magnetic current density at both exterior and interior boundaries, respectively. The magnetic current density is specified as a three-dimensional vector, but because it needs to flow along the boundary surface, COMSOL Multiphysics projects it onto the boundary surface and neglects its normal component. This makes it easier to specify the magnetic current density and avoids unexpected results when a magnetic current density with a component normal to the surface is given.

Enter values or expressions for the components of the **Surface magnetic current density** $\mathbf{J}_{\text{ms0}}$ (SI unit: V/m).

Symmetry Plane (for Magnetic Flux Density)

The **Symmetry Plane** node adds a boundary condition that represents symmetry or antisymmetry in the magnetic flux density, depending on which option is chosen.

SYMMETRY TYPE

Choose between **Symmetry** and **Antisymmetry** for the magnetic flux density. If **Symmetry** is chosen, the boundary condition is

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

which states that the normal component of the magnetic flux density is zero. If **Antisymmetry** is chosen, the boundary condition becomes

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

which states that the tangential components of the magnetic flux density are zero.

	Magnetotellurics: Application Library path **ACDC_Module/ Devices,_Resistive/magnetotellurics**

Magnetic Potential

By default, the **Magnetic Potential** node adds a boundary condition for the magnetic vector potential:

$$\mathbf{n} \times \mathbf{A} = \mathbf{n} \times \mathbf{A}_0$$

where $\mathbf{A}_0$ is defined in the **Magnetic Potential** section. As the equation suggests, the boundary condition only guarantees that the tangential components of $\mathbf{A}$ and $\mathbf{A}_0$ are equal. Their normal components might deviate depending on specific situations. For reduced field formulation where the background field is specified in the interface settings, the node acts the same as the Magnetic Insulation feature.

Enter a value or expression for the **Magnetic vector potential $\mathbf{A}_0$** (SI unit: Wb/m) coordinates.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

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.
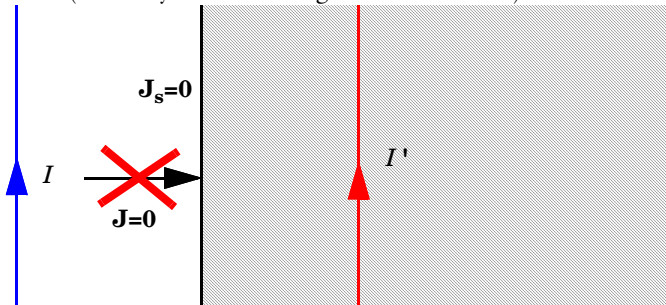
Figure 5-2: The perfect magnetic conductor boundary condition is used on exterior boundaries representing the surface of a high impedance region or a symmetry cut (equivalent to using antisymmetry in Symmetry Plane feature). The shaded (high impedance) region is not part of the model but carries effective 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.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

Periodic Pair

Use the **Periodic Pair** node, selected from the **Pairs** menu, to specify a pair where the periodic condition is applied to the magnetic vector potential. It is used to constrain continuity in a linear periodic machine where only a finite part is modeled. This node is different from the Periodic Condition feature that does not apply to pairs.

PAIR SELECTION

From the **Pairs** list, choose **From list** (the default), **All**, **All contact pairs**, or **All identity pairs**. If you choose **From list**, select the pairs where you want to impose continuity across the pair boundaries by clicking the **Add** button (+) and choose the pairs to add from the list in the **Add** dialog that opens; then click **OK**.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

▥	*Linear Motor in 2D*: Application Library path **ACDC_Module/ Devices,_Motors_and_Generators/linear_motor_2d**

Line Current (Out-of-Plane)

Use the **Line Current (Out-of-Plane)** node, selected from the **Points** menu, to specify a line current out of the modeling plane. In axially symmetric geometries this is the rotational (azimuthal) direction; in 2D geometries this is the z direction.

LINE CURRENT (OUT-OF-PLANE)

Enter a value or expression for the **Out-of-plane current** I_0 (SI unit: A).

Line Current (on Axis)

The **Line Current (on Axis)** feature is available for 2D axisymmetric components. It applies to edges that coincide with the symmetry axis, and is active only when in-plane magnetic vector potential components are available (for more information, see Components). Use the **Line Current (on Axis)** feature to specify a line current along the symmetry axis.

Enter a value or expression for the **Line current in z direction** I_z (SI unit: A).

Electric Point Dipole

Add **Electric Point Dipole** nodes to 3D and 2D components. This represents the limiting case of when the length d of a current filament carrying uniform current I approaches zero while maintaining the product between I and d. The dipole moment is a vector entity with the positive direction set by the current flow.

DIPOLE SPECIFICATION
Select a **Dipole specification** — **Magnitude and direction** (the default) or **Dipole moment**.

DIPOLE PARAMETERS
Under **Dipole specification**:

- For **Magnitude and direction** enter coordinates for the **Electric current dipole moment direction** $\mathbf{n}_p$ (the defaults are 0) and **Electric current dipole moment, magnitude** p (SI unit: Am) (the default is 0 Am).

- For **Dipole moment** enter coordinates for the **Electric current dipole moment** $\mathbf{p}$ (SI unit: Am). The defaults are 0 Am.

Gauge Fixing for A-Field

For 2D and 2D axisymmetric components. 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 in the **Settings** window for The Magnetic Fields Interface.

The **Gauge Fixing for A-Field** node enforces the gauge $\nabla \cdot \mathbf{A} = 0$ 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**.

<table>
<tr>
<td>🔍</td>
<td>

See two blog posts about what is and how to use the gauge fixing:

- https://www.comsol.com/blogs/
  what-is-gauge-fixing-a-theoretical-introduction/
- https://www.comsol.com/blogs/
  how-do-i-use-gauge-fixing-in-comsol-multiphysics/

</td>
</tr>
</table>

DOMAIN SELECTION

From the **Selection** list, choose the domains to define the gauge-fixing potential ψ. In most cases, the feature should be applied to all domains where the magnetic vector potential **A** is solved for. By default, the selection is set to **All domains**, ensuring that the gauge fixing is applied to all the valid domains in the model.

GAUGE FIXING FOR A-FIELD

In 3D, there is an option to change the feature's **Equation form**, which is set to **Study controlled** by default. For Time Dependent and Frequency Domain studies, set **Equation form** to **Stationary** can be efficient to handle regions without conductivity.

The variable ψ is used to impose a condition on the derivatives of the magnetic vector potential, so its absolute value does not have particular significance; only its gradient enters the equations (the variable ψ acts more or less like a potential). The absolute value of the variable can be set by entering the **Divergence condition variable scaling** ψ_0 (SI unit: A/m). The default value is 1 A/m, which is appropriate for most models.

ADVANCED SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

This section allows a more fine control on the boundary conditions for ψ applied by the **Gauge Fixing** feature. The domain equation for ψ only imposes a condition on the gradient, so it is important to constrain the absolute value of ψ to ensure a nonsingular model. The **Constant value on insulation boundaries** checkbox (selected by default) imposes a constant value on the conductive boundaries in the model, such as Magnetic Insulation.

If there are no such boundaries, select the **Ensure constraint on value** to ensure that there is always a constraint set (at least in one point) on the value of the divergence condition variable ψ.

> ⚠️ If you apply **Gauge Fixing** across pair boundaries, the internal logic for constraining ψ in at least one point does not work. The problem is most likely to appear in gauge fixing for rotating machinery. Assume you have one gauge fixed **Ampère's Law** domain in the stator and one in the rotor, then one must use separate **Gauge Fixing** features for these as the geometry analysis for setting up point constraints on ψ does not work across pairs. It can also be handled by adding manual constraints on ψ.

> ⚠️ Applying **Gauge Fixing** to nonlinear models, for example with materials involving magnetic saturation has the known side-effect that the nonlinear solver fails to converge for models with zero or numerically small excitation. Thus, starting for example parametric sweeps from exactly zero applied source currents should be avoided.

Magnet

The **Magnet** node is used to model one or multiple magnets, simplifying the definition of the magnetization direction in each magnet. In configurations with electric conductivity, this feature allows for imposing electrical insulation on boundaries, making it easier to compute losses in segmented magnets. It is also possible to add a Loss Calculation subfeature in order to compute induced losses. In The Rotating Machinery, Magnetic Interface, this feature is called **Conducting Magnet**.

MAGNET

Direction Method
Select the **Direction method** to be used for determining the direction of the magnetization in the magnet. Choose between:

- **Specify north and south boundaries**, where the north and south boundaries of the magnet can be selected. The direction of the magnetization is then calculated based on the position average of each pole.
- **Specify by domains**, which allows the magnetization direction to be set individually in different subselections of the **Magnet** feature. The magnetization in each such

domain can then be set by either specifying the north and south boundaries, or by entering a user defined direction.

- **User defined**, which allows the user to manually specify a direction vector for the magnetization.

Pattern Type

This section is only available when **Specify north and south boundaries** is selected as the **Direction method**. The setting makes it possible to automatically set the magnetization direction for multiple magnets following a certain pattern. Choose between **No pattern**, **Circular pattern**, and **Linear pattern (or domain index based)**. When **Circular pattern** is selected, the magnetization direction for each pattern element is determined by its position in a circle and a pattern repetition angle. Select **Linear pattern (or domain index based)** to repeat the definition of the magnetization direction in a selection of magnets that are linearly displaced.

Type of Periodicity

This section is only available when **Circular pattern** or **Linear pattern (or domain index based)** is chosen as the **Pattern type**. The different options are **Repeating**, **Alternating**, and **User defined**. When **Repeating** is selected, the magnetization direction is the same in each element. Select **Alternating** to instead have the magnetization direction reversed for each repeated element in the pattern. If the alternating pattern is applied to an odd number of magnets, the north and south boundaries should be specified for the first magnet in the pattern. The **User defined** option is only available for a circular pattern and allows a custom input that determines how much the magnetization direction should change between each element.

Pattern Additional Angle

This section is only available when **User defined** is chosen as the **Type of periodicity**, when also having **Circular pattern** as the **Pattern type**. This angle is added to the magnetization direction for each repetition in the pattern.

Pattern Center

This section is only available when **Circular pattern** is chosen as the **Pattern type**. It defines the center of the circular pattern.

Pattern Axis

This section is only available when **Circular pattern** is chosen as the **Pattern type**. This setting is only available in 3D, and defines the axis around which the pattern is created.

Direction of Magnetization

This section is only available when **User defined** is selected as the **Direction method** in the **Magnet** feature. This input vector is equivalent to the one that can be added when **Magnetization** is selected in the Ampère's Law feature.

CONSTITUTIVE RELATION B-H

This section is identical to the one in the Ampère's Law node, apart from that only the options **Remanent flux density** and **Nonlinear permanent magnet** are available here.

CONSTITUTIVE RELATION JC-E

This first part of this section is identical to the one in the Ampère's Law node, when **Electric conductivity** is chosen as the **Conduction model**. There is also an additional setting here, called **Constrain for induced currents**, that determines which boundaries are electrically insulated. The different options are **Within each domain**, **No induced current constrain**, **Within each connected component**, and **Within all domains**. Selecting **Within each domain** electrically insulates each selected domain separately, and constrains the currents within. Choosing **No induced current constrain** places no constraint on the induced currents. The option **Within each connected component** instead constrains the currents in each subselection of domains that are geometrically connected. For example, two adjacent domains will be electrically insulated around their outer common boundary, while for domains that are separated the effect will be the same as when selecting **Within each domain**. When selecting **Within all domains**, the net current of all selected domains will be constrained to zero. This setting is only available in 2D, and can be thought of as all of the domains being electrically connected outside of the plane. With this setting there can be a nonzero net current in some domains, but it will be equalized by currents in the opposite direction in other domains.

North

The **North** subnode is added by default as a subnode of the **Magnet** node, whenever **Specify north a south boundaries** is chosen as the **Direction method**. It is also added as a subnode to a **Domain Direction** node with the same setting. Use this feature to select the exterior boundaries that should be the north pole boundaries of the magnet.

South

The **South** subnode is added by default as a subnode of the **Magnet** node, whenever **Specify north and south boundaries** is chosen as the **Direction method**. It is also added as

a subnode to a **Domain Direction** node with the same setting. Use this feature to select the exterior boundaries that should be the south pole boundaries of the magnet.

Domain Direction

The **Domain Direction** subnode is added by default as a subnode of the **Magnet** node, whenever **Specify by domains** is chosen as the **Direction method**. This node contains the same two settings as its parent feature, **Specify north and south boundaries** and **User defined**, which will be applied to its selection. Multiple copies of this feature can be added under one **Magnet** feature in order to individually specify the magnetization direction of different domains.

Passive Conductor

The **Passive Conductor** node is a simplified version of the Magnet feature. It is used to model nonmagnetized conductors with thin interior or exterior electric insulation boundaries. This is useful for the fully resolved modeling of eddy currents in segmented or layered conductors. For thin iron laminates, consider modeling the material as an *effective medium* and use the **Steinmetz** or **Bertotti** loss models supplied by the Loss Calculation subfeature.

CONSTITUTIVE RELATION B-H

This section is identical to the one in the Ampère's Law node, except for that only the options **Relative permeability**, **B-H curve**, and **Effective B-H curve** are available here.

CONSTITUTIVE RELATION JC-E

This section is identical to the one in the Magnet node.

Laminated Core

The **Laminated Core** node is used to model an anisotropic effective material for laminated soft magnetic cores that are not developing induced currents. The **Magnetization model** can be set to **B-H Curve** (default), **Effective B-H Curve**, and **Relative Permeability**, which have the same settings as described in the Ampère's Law node of the Magnetic Fields interface. The **Lamination model** is used to choose between **Homogeneous** (default), **Out-of-plane** (available in 2D only), and **Direction based** types of lamination. The first is used to model uniform material cores, as described in the Ampère's Law node of the Magnetic Fields interface. The latter are used to reproduce the homogenized effect of a stack of soft iron laminates and insulation spacers along a

direction of lamination, based on the Energy- and Coenergy-Based Magnetic Constitutive Relations. The direction of lamination and the stacking factor (the volumetric ratio between the laminates and the total stack) can be set. These two quantities are used to evaluate the effective magnetic energy density of the stack and the corresponding effective magnetic field.

It is also possible to specify the **Loss Model**. When enabled, it allows to define the **Volumetric loss density** (SI unit: W/m^3) and it can be specified as **User defined**, or, if **B-H Curve** is specified, by using empirical loss formulas via **Steinmetz** or **Bertotti** coefficients. More information on loss models can be found in the Loss Calculation node documentation. A **Stabilization** section is also available to improve numerical stability. It is described under Stabilization of the **Free Space** node. In the The Rotating Machinery, Magnetic Interface, this feature is called Laminated Core, Ampère's Law.

Coil

The **Coil** node can be used to model coils, cables, and other conductors subject to a lumped excitation, such as an externally applied current or voltage. The Coil feature transforms this lumped excitation into local quantities (electric field and electric current density), and computes lumped parameters of interest such as impedance, and inductance.

The **Coil** feature supports three different **Conductor models**:

- **Single conductor**, which models a conductive body such as a wire, busbar, or other metallic object in which the current flows freely due to the material's conductivity. This model can be used when the current flow has a well-defined beginning and end (for example, connections to an external source) or is closed in a loop.

- **Homogenized multiturn**, which models a bundle of electrically thin wires tightly wound together but separated by an electrical insulator. In this scenario, the current flows only in the direction of the wires. Induced currents flowing in other directions are considered negligible.

- **Homogenized litz coil**, which is a variant of the multiturn model where the effects of helical twisting and multiple strands per turn are accounted for.

!	In 3D, the **Domain Selection** has to be complete in the sense that selecting only part of a contiguous conductor will lead to unphysical results. If the current loop is not complete, Ampère's law may be violated.

The **Coil** feature is available both for domain and for boundary selections. In the latter case, it represents a flat coil or a conductor with a thickness negligible compared to the other dimensions. Various subnodes can be added to the **Coil** node in certain cases.

The global Harmonic Perturbation subnode is available from the context menu (right-click the parent node and select it from the **Global** menu) or from the **Physics** toolbar, **Attributes** menu. The subnode can be used to apply a harmonic perturbation to the coil excitation.

In 2D and 2D axisymmetric components, the **Coil** feature supports the **Coil group** functionality. This can be activated by selecting the corresponding checkbox.

⚠️ The **Coil group** option assumes that the selected domains represent cross sections of the same conductor going in and out of the modeling plane. These domains are expected to have the same areas. The same total current will be imposed in each domain, even if the domain areas are not equal. If the areas are unequal, the computed concatenated flux, coil voltage and inductance will be incorrect. For cases with varying cross section areas, it is recommended to use separate coil features that are coupled using The Electrical Circuit Interface.

Refer to the Modeling Coils section in the modeling guide for more information about this node.

In 2D and 2D axisymmetric components, the wires are assumed to be in the out-of-plane direction.

🔍 See Modeling Coils in this guide to learn more about using this feature.

MATERIAL TYPE

The **Material type** setting decides how materials behave and how material properties are interpreted when the mesh is deformed. Select **Solid** for materials whose properties change as functions of material strain, material orientation and other variables evaluated in a material reference configuration (material frame). Select **Nonsolid** for materials whose properties are defined only as functions of the current local state at

each point in the spatial frame, and for which no unique material reference configuration can be defined. Select **From material** to pick up the corresponding setting from the domain material on each domain. Since **Coil** features model conductors or bundles of wires, the correct choice is usually **Solid**.

COIL

Coil Name
Enter a **Coil name**. This name is appended to the global variables (current, voltage) defined by this coil, and it can be used to identify the coil in a **Coil Geometry Analysis** study step.

Conductor Model
Select the **Conductor model** for the coil. The choices correspond to rather different physical models, although the setup is similar. The **Single conductor** model (the default) is appropriate for fully resolved conductors with skin and proximity effects included. The **Homogenized multiturn** model represents a bundle of electrically thin wires that are not geometrically resolved but included as an *effective medium*. Skin and proximity effects can be included in a lumped sense, using the **Harmonic loss** effective loss model or a **Mutually coupled circuit**. The **Homogenized litz coil** is a variant of the multiturn model where the effects of helical twisting and multiple strands per turn are accounted for. The choice of conductor model affects the controls that are visible in the GUI and the available subnodes for the Coil feature.

Coil Type (3D, Homogenized Multiturn, Homogenized Litz Coil)
This section is available when selecting **Homogenized multiturn** or **Homogenized litz coil** as the **Conductor model** in 3D, and is used to specify the coil geometry (the orientation of the wire bundle).

Select a **Coil Type** — **Numeric** (the default), **Circular**, **Linear**, or **User defined**. The different alternatives are described in the following sections. Also see Using Coils in 3D Models for more information.

	An additional subnode is needed to completely define the coil geometry:
!	• For **Numeric** the Geometry Analysis subnode is added by default.
	• For **Circular** or **Linear** the Coil Geometry subnode is added by default.
	• For **User defined** the User Defined Coil Geometry subnode is added by default.

When **Single conductor** is selected as the **Conductor model**, the coil behaves as if **Coil Type** is **Numeric**, including the presence of the Geometry Analysis subnode.

Numeric Coil Types

In a **Numeric** coil, the current flow is computed automatically in a **Coil Geometry Analysis** study step. Use the Geometry Analysis subnode to set up the problem.

ⓠ	Automatic Coil Geometry Analysis

Circular Coil Types

In a **Circular** coil, the wires are wound in circles around the same axis. Use the Coil Geometry subnode to select a group of edges forming a circle or a part of a circle around the coil's axis. From the selected edges, the coil axis is computed and the direction of the wires is taken to be the azimuthal direction around the axis, as marked by the red arrows. The coil length used is computed as the coil volume divided by the coil cross sectional area, unless the **Use robust geometry analysis method** box is checked. When the robust method is used, the coil length is simply the length of the selected edges.

Linear Coil Types

In a **Linear** coil, all the wires are parallel and straight lines. Use the Coil Geometry subnode to select an edge or a single group of connected edges that maps out the local coil direction. The direction of the wires and the coil length is taken to be the direction and the length of the edge(s), as marked by the red arrow. Avoid selecting multiple parallel edge groups as that will result in an incorrect coil length.

To respect the current conservation law, the applied currents cannot originate from interior boundaries. A Linear coil should therefore be terminated on exterior boundaries.

User Defined Coil Types

For **User defined** manually specify the direction of the wires as a vector field and the length of the coil. Use the User Defined Coil Geometry subnode to specify the coil geometry.

Coil Group (2D and 2D Axisymmetry)

The **Coil group** checkbox is only available for 2D and 2D axisymmetric components. Select this checkbox to enable the **Coil group** mode for this feature. With this setting, the domains or domain groups in this feature's selection are considered

series-connected. Selecting this checkbox activates the **Domain Group** subnode. See Coil Groups for more information.

Coil Excitation

Select a **Coil excitation** — **Current** (the default), **Voltage**, **Circuit (voltage)**, **Circuit (current)**, or **Power** (2D and 2D axisymmetric components only).

- **Current** forces a total current flowing in the coil wire. Enter a **Coil current** I_{coil} (SI unit: A). The default is 1 A. See the box below for study limitations on this setting.

- **Voltage** applies a total voltage across the coil enter a **Coil voltage** V_{coil} (SI unit: V). The default is 1 V.

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

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

- **Power** (only available for 2D and 2D axisymmetric components) forces the coil input power (cycle-average in frequency studies) to the specified value. Choosing this option makes the problem nonlinear. For **Power** enter a **Coil power** P_{coil} (SI unit: W). The default value is 1 W.

!	When using the **Current**, **Circuit (current)**, and **Power** options, the coil feature sets up a control problem for the coil voltage and current. Due to its complexity, the following limitations apply:
	- The **Power** option is only available for 2D and 2D axisymmetric components.
	- The **Current** and **Circuit (current)** options should be used with care in **Time Dependent** study steps. One should avoid applying a current step excitation (for example a fixed nonzero current) as that will lead to unphysical results and/or numerical instability.
	- See Modeling Coils in this guide to learn more about using this feature.

HOMOGENIZED CONDUCTOR

This section is available only when **Homogenized multiturn** or **Homogenized litz coil** is selected as the **Conductor model**. In this case, the coil represents a bundle of electrically thin wires separated by an insulating material. Additional settings can be specified.

Number of Turns

Enter the **Number of turns** N. The default is 10. This is the number of wires constituting the coil cross-section. The coil resistance is affected by this number and so is the current density in the coil. Together with the **Current** setting the number of turns defines the number of Ampère-turns in the coil.

Wire Properties

The **Wire properties** can be specified using the following options:

- **From conductivity** allows for specifying the coil wire conductivity σ and a coil wire cross-section area. The area needs to be the area of the actual conductor, without varnish or insulation. Several standards are supported.

- **From resistivity**; specify the wire resistance per unit length ρ_{wire}. This option is typically used for litz wires, where the resistance per unit length is taken from measurements or a specification sheet provided by the supplier.

- **From resistance**; specify the total wire or coil resistance R_{coil}.

- **From resistance and mutually coupled circuit**; specify the coil resistance R_{coil}, the mutual inductance M_{cir}, and the source of the mutually coupled circuit current I_{cir}. If the latter is set to **Series R and L**, also specify the mutually coupled circuit resistance R_{cir} and the mutually coupled circuit inductance L_{cir}. If it is set to **User defined**, just specify I_{cir} directly.

> ⚠️ *Note that the AC resistance does not have a unique interpretation.*
>
> The **Wire properties** are used to determine the *intrinsic* resistance of the coil wire or litz wire. Usually, this amounts to the *DC resistance*. If a frequency-dependent *AC resistance* is used — for instance from a litz wire specification sheet — it will include both the DC resistance and additional resistance from loss terms caused by skin and proximity effects *within* the strand bundle that forms the litz wire. This intrinsic AC resistance is a property of the (litz) wire itself.
>
> If the wire is then wrapped around a magnetic core with a nonzero conductivity it will lead to additional eddy currents and additional loss. The *total loss* will determine the final coil *AC resistance* as perceived by the power source. Depending on the coil geometry, core type, and nearby conductors this resistance may be different from the AC resistance used to specify the wire properties.

Litz Wire DC Resistance and Number of Strands (Conductivity)
Available for the **Homogenized litz coil** conductor model only. The **Litz wire DC resistance per unit length** ρ_{dc} is typically around 3–10% higher than the DC resistance of a bundle of untwisted strands with the same bundle length and cross sectional copper content. This is because the helical path length of the individual strands is a bit longer than the bundle length. The effect can be compensated for by specifying the DC resistance as measured or as provided by the litz wire specification sheet. When unknown, an increase of 6% is a fair estimate.

Enter the **Number of strands** n as included in the litz wire. This parameter is used together with the **Strand conductivity**, and the **Strand area** to determine the untwisted DC resistance and the copper packing density.

Coil Wire Conductivity / Wire Strand Conductivity (Conductivity)
Enter a **Conductivity** σ (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 and will be used for determining the lumped resistance and the loss. Note that this is not the bulk conductivity of the material as perceived by free eddy currents, which is instead set to a very low value comparable to the **Free space** stabilization conductivity. Intrinsic parasitic inductive effects such as skin and proximity effects in the coil itself are instead modeled in a lumped fashion, using the **Harmonic loss** effective loss model or a **Mutually coupled circuit**.

Coil Wire Cross-Section Area / Wire Strand Cross-Section Area (Conductivity)
Enter the cross-section area of the individual wire or strand in the coil. It is used, for example, to compute the lumped resistance of the coil. The area can be specified in different ways, according to the option selected in the **Coil wire cross-section area** list — **Filling factor** (the default), **From diameter**, **Standard wire gauge**, **American wire gauge (Brown & Sharpe)**, or **User defined**.

- For **Filling factor**, enter the filling factor f. This dimensionless factor sets the fraction of the cross-section occupied by the metal (usually copper), also known as the *packing density*. Use this setting to specify the wire or strand area when the area is unknown or is yet to be determined, and the coil geometry is a reasonably accurate representation of the real device. The value is expected to be below 0.9, which is the densest packing for circles. The option is not available for boundary coils.

- For **From diameter**, enter the diameter of the individual wire or strand d (SI unit: m). The cross-section area of the round wire will be computed from it. The default value of d is 1 mm.

- For **Standard wire gauge**, enter the SWG size. Sizes between 7/0 and 50 are available. The default size is 0.

- For **American wire gauge (Brown & Sharpe)**, enter the AWG size. Sizes between 0000 and 40 are available. Sizes such as 0000 can be also written as 4/0. The default size is 0.

- For **User defined**, enter the value of the cross section area a (SI unit: m^2). The default is 1 mm^2.

⚠️ *Note that the packing density may surpass 0.9.*

When using the options **From diameter**, **Standard-** or **American wire gauge**, or **User defined**, there is a risk of the filling factor passing the 0.9 threshold unnoticed. If the coil geometry in the model is an accurate representation of the real device (not an approximate prototype or a proof of concept) this may indicate a design flaw or a modeling error; *the coil should not contain more metal than physically possible*. Higher values are allowed, however, for experimentation, sweeping, or optimization purposes. Use the coil filling factor variable in postprocessing to verify coil integrity.

High-Frequency Effective Loss (Conductivity)

For domain coils where the wire or strand properties are specified using the conductor area and wire conductivity, the **Include harmonic loss** option is available for computing the coil's intrinsic AC loss in the frequency domain (enabled by default). For stationary and transient studies, it reverts back to DC behavior — for including intrinsic AC loss in a Transient analysis, consider using the **Mutually coupled circuit option**.

When the frequency, the wire or strand size (and distance) with respect to the skin depth, the fill factor and the wire conductivity are known, it is possible to include intrinsic AC losses in the coil (or litz wire) based on a *continuum skin and proximity effect model*, see Ref. 1. At frequencies where the skin depth approaches the strand size, the intrinsic loss starts to dominate and the resistance of the coil can become orders of magnitude larger than the DC resistance.

❗ Note that in order to get realistic resistive and inductive behavior, the **High-frequency effective loss** model not only specifies a frequency dependent AC resistance, but also enforces an effective complex permeability. The relative permittivity is assumed to be one.

> ⚠ Note that for the harmonic loss model to produce accurate results, the filling factor should have a realistic value for a bundle of hexagonally close packed strands (typically in the range of 0.5–0.9). Use the coil filling factor variable in postprocessing to verify coil integrity.

Wire Resistance Per Unit Length (Resistivity)

Specify a **Wire resistance per unit length** ρ_{wire} in Ω/m. The default value is $30\ \Omega$ per 1000 feet. This option is typically used for litz wires, where the resistance per unit length is taken from measurements or a specification sheet provided by the supplier. Other common sources are analytic litz wire models (that can be typed directly into the expression field for ρ_{wire}) or 2D finite element models that fully resolve the strands and output the frequency-dependent AC resistance in the form of a lookup table.

Coil Resistance (Resistance)

Only available for the **Homogenized multiturn** conductor model. Specify the total **Coil resistance**. This will typically be the total DC resistance. When a frequency dependence is included in the expression, it will be the *intrinsic* AC resistance — For more information, see Wire Properties. The default value is $50\ \Omega$.

Mutually Coupled Circuit Loaded Homogenized Multiturn

Only available for the **Homogenized multiturn** conductor model. When the **From resistance and mutually coupled circuit** option is chosen, the coil resistance R_{coil} and the inductance will approach the DC values in the low frequency regime. When the frequency increases, the magnetic fields are still described by Ampère's law where the integrated current density in each coil section is the coil current multiplied by the number of turns. However, the presence of I_{cir} will affect the total potential that generates the coil current as an extra load opposed to the main coil current.

This may be seen as an effect of the interaction between the outermost and innermost parts of the current density in each coil turn. Therefore, with properly tuned values of R_{cir} and L_{cir}, it is possible to mimic the increased resistance due to skin and proximity effects between the individual turns in the multiturn coil. Finding these values is typically done by fitting the parameters (using a sweep, or optimization) such that the coil shows the same time-average behavior as a fully resolved 2D model.

Expressing this in equations, we have the finite elements inductive effect on the electric potential

$$V_{\mathrm{MAG}} \ = \ -\int \frac{d\mathbf{A}}{dt} \cdot d\mathbf{s} \ = \ -N_{\mathrm{coil}} \int \mathbf{e}_{\mathrm{coil}} \cdot \frac{d\mathbf{A}}{dt} dV$$

where the first integral is along the coil wire and the second is over the coil volume. Assuming a default choice of I_{cir} that balances R_{cir} and L_{cir}, we then get

$$-V_{\text{coil}} + V_{\text{MAG}} - R_{\text{coil}}I_{\text{coil}} + M_{\text{cir}}\frac{dI_{\text{cir}}}{dt} = 0$$

$$-R_{\text{cir}}I_{\text{cir}} - L_{\text{cir}}\frac{dI_{\text{cir}}}{dt} + M_{\text{cir}}\frac{dI_{\text{coil}}}{dt} = 0$$

for the coil and the mutually coupled current loop.

SYMMETRY SPECIFICATION (2D AND 2D AXISYMMETRIC COMPONENTS)
For domain coils in 2D and 2D axisymmetry, symmetry factors can be entered at the coil's main level (note that for coils in 3D these factors are typically set in the **Geometry Analysis** subnode). If the model represents only a part of the full device because of the use of symmetry, these factors can be used to compensate for parts of the coil missing. Attached circuits and lumped quantities such as the inductance or the resistance will then behave as if the model represents the full device.

Enter the **Coil length multiplication factor** F_L and **Coil area multiplication factor** F_A (dimensionless integer numbers) to compensate for the difference in turn length, and the difference in conductor (or bundle) thickness.

To model a fully resolved litz wire (or litz coil) in 2D, add all the strands as individual circles in the geometry and mesh. Use a **Single conductor** coil to fully resolve the skin and proximity effects, and use a coil group to put all the strands in series — *enforcing the same current in each of them.* Then, set F_L equal to $1/n$ and F_A equal to n, where n is the number of strands. This will effectively turn the series connection into a parallel connection, while still enforcing the same current in all strands. In order to model a litz coil without resolving the strands, use the **Homogenized litz coil** effective conductor model, and enable **Include harmonic loss**.

CONSTITUTIVE RELATION JC-E
This section is available only for the **Single conductor** coil model. It allows to set the resistive bulk properties. In this case, the coil represents a solid (fully resolved) conductor and the conductivity of the material is required to compute the current density flowing in it. This section is identical to the one in the Ampère's Law node.

CONSTITUTIVE RELATION B-H, AND D-E

For a domain coil, the **Constitutive Relation B-H** and **Constitutive Relation D-E** sections can be used to specify the magnetic and dielectric bulk properties of the medium. When the **Single conductor** model is selected, these options are essentially the same as those used in the Ampère's Law node.

When the **Homogenized multiturn** or **Homogenized litz coil** model is selected, the number of options is restricted to better match the assumptions that the effective material model is based on; a densely packed strand bundle with magnetic and dielectric bulk properties close to those of **Free space**. The chosen settings now represent the average properties of both the wires and the insulation, or epoxy resin.

STABILIZATION

To display this section, click the **Show More Options** button (👁) and select **Stabilization** in the **Show More Options** dialog. This section is available only in 3D components when using **Homogenized multiturn** as the **Conductor model** and it contains advanced settings relative to the accuracy and stabilization of the solution.

The **Accurate coil voltage calculation** checkbox enables a current filtering functionality that improves the accuracy of the computed electric field and the induced coil voltage, at the cost of a slightly increased number of degrees of freedom. This functionality is only applicable for time dependent and frequency domain studies, and is active by default.

For the purpose of stabilizing the solution, the coil feature can apply a small electric conductivity to the coil domain. Use the **Stabilization** combo box to specify the value of the conductivity. Choose **Automatic** (the default) to use a conductivity automatically computed by the coil. In frequency domain studies, the conductivity is chosen so that the skin depth in the coil is much larger than the coil length (see the sections Coil Geometry and User Defined Coil Geometry below). It is deduced from the formula

$$\sigma_{\text{stab}} = 0{,}001\left(\frac{2}{\omega\mu\delta^2}\right)$$

by setting the skin depth δ, equal to the coil length. In other study types the conductivity is set to 1 S/m.

If **None** is chosen, no conductivity is used in the coil domain. Choose **User defined** to specify the **Electric conductivity in the coil domain** σ_Ω (SI unit: S/m). The default value is 1 S/m. The purpose of this electric conductivity is only to stabilize the solution.

According to the Homogenized multiturn model, the domain should not be conductive and all the currents should flow in the direction of the wires only.

- Coil Excitation
- Power Excitation
- Using Coils in 3D Models

Harmonic Perturbation

The **Harmonic Perturbation** subnode is available at the global level from the context menu (right-click the Coil parent node) or from the **Physics** toolbar, **Attributes** menu. Use this feature to specify a harmonic perturbation for the coil excitation.

COIL
A harmonic perturbation can be specified only if the **Coil excitation** of the parent feature is **Coil current** or **Coil voltage**. In the first case, enter a **Coil current** I_{coil} (SI unit: A). The default is 1 A. In the second case, enter a **Coil voltage** V_{coil} (SI unit: V). The default is 1 V.

Harmonic Perturbation, Prestressed Analysis, and Small-Signal Analysis in the *COMSOL Multiphysics Reference Manual*

Coil Geometry

The **Coil Geometry** subnode of the Coil is available if the **Conductor model** of the parent feature is **Homogenized multiturn** and the **Coil type** is **Linear** or **Circular**. This feature was previously called **Reference Edge**. This subnode is used to specify the direction of the tiny wires constituting the multiturn coil.

- In **Linear** coils, 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 edges. To avoid unphysical currents, a **Linear** coil should be terminated on external boundaries.

- In **Circular** coils, the wires are wound in circles around the same axis. Select a group of edges forming a circle or a part of a circle around the coil's axis. From the selected edges, 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 computed as the coil

volume divided by the coil cross-sectional area, unless the **Use robust geometry analysis method** box is checked. When the robust method is used, the coil length is simply the length of the selected edges.

Select the **Use robust geometry analysis method** checkbox to use an alternative algorithm for the computation of the coil axis and direction. This method works even in models without geometry (for example, with an imported mesh), but requires that the selected edges form a complete circle.

COIL GEOMETRY

- The direction of the coil is shown in the **Graphics** window with a red arrow. Select the **Reverse direction** checkbox to reverse the direction of the coil.

- The length of the coil is computed from the length of the selected edges. To manually specify the length, select the **Override length of the edges** checkbox, and enter the **Total length of the edges** L_{edges} (SI unit: m).

- In domain features, the average cross-section area of the domain is computed automatically from the domain volume and the length of the edges. The cross-section area is used, for example, to compute the coil current density. To manually specify a cross-section area, select the **Override domain area** checkbox, and enter the **Average domain cross section area** A_{domain} (SI unit: m^2)

- If the model represents only a part of a geometry obtained from symmetry cuts (for example, a quarter of a circle), use the settings under **Symmetry specification** to specify appropriate correction factors. Enter the **Coil length multiplication factor** F_L and **Coil area multiplication factor** F_A (dimensionless integer numbers). The actual length of the coil, used to compute the coil voltage and resistance, is then computed as the product $F_L \cdot L_{\text{edges}}$. The cross-section area of the coil is computed as $F_A \cdot A_{\text{domain}}$.

Q

- See Using Coils in 3D Models in the modeling section to learn more about this node.

- The Magnetic Fields Interface

- Theory for the Magnetic Fields Interface

User Defined Coil Geometry

The **User Defined Coil Geometry** subnode is available for Coil if the **Conductor model** of the parent feature is **Homogenized multiturn** and the **Coil type** is set to **User defined**.

Specify in this feature an arbitrary vector field that defines the direction of the wires in the coil. Such field can be an analytic expression or can be computed using, for example, a **Curvilinear Coordinates** physics.

For domain features, information on the input/output boundaries is necessary to complete the setup of the coil feature. Use the **Input** and **Output** subfeatures to this node to select the input/output boundaries of the coils, external boundaries where the vector field representing the wire direction has a nonzero normal component. The selection can be left empty if there are no such boundaries, for example in the case of a closed-loop coil. This operation is necessary in time dependent and frequency domain studies if the **Accurate voltage calculation** is active in the parent feature.

COIL GEOMETRY

- Enter the **Coil current flow** $\mathbf{e}_{\mathrm{coil}}$, a vector field representing the local direction of the wires, which can be an analytic expression, or the solution of another physics (for example, Curvilinear Coordinates). The vector field will be normalized by dividing the local field by the volume average of the field norm. This keeps local variations intact, while scaling the average norm to unity.

- Enter the **Domain length** L_{domain} (for domain features) or the **Boundary length** L_{boundary} (for boundary features). The SI unit is meters. This value should correspond to the average length of the wires in the coil, before any symmetry correction.

- If the model represents only a part of a geometry obtained from symmetry cuts (for example, a quarter of a circle), use the settings under **Symmetry specification** to specify appropriate correction factors. Enter the **Coil length multiplication factor** F_L and **Coil area multiplication factor** F_A (dimensionless integer numbers). The actual length of the coil, used to compute the coil voltage and resistance, is then computed as the product $F_L \cdot L_{\mathrm{domain}}$ (or $F \cdot L_{\mathrm{boundary}}$ for boundary features). The cross-section area of the coil is computed as $F_A \cdot A_{\mathrm{domain}}$.

Input (for User Defined Coil Geometry)

The **Input** subnode is added by default as subnode of the User Defined Coil Geometry node, and is used to provide more information on the coil geometry. Use it to specify the boundaries where the wires (the vector field specified in the parent User Defined Coil Geometry feature) enter the domain. The selection can be left empty in case of a closed-loop coil.

This node should be used in combination with the Output (for User Defined Coil Geometry) node.

Output (for User Defined Coil Geometry)

The **Output** subnode is added by default as subnode of the User Defined Coil Geometry node and is used to provide more information on the coil geometry. Use it to specify the boundaries where the wires (the vector field specified in the parent User Defined Coil Geometry feature) leave the domain. The selection can be left empty in case of a closed-loop coil.

This node should be used in combination with the Input (for User Defined Coil Geometry) node.

Override Coil Inlet/Outlet

The **Override Coil Inlet/Outlet** subnode is available at both the domain and boundary level from the context menu (right-click the Coil parent node) or from the **Physics** toolbar, **Attributes** menu. This is an advanced feature, and is normally not required.

This feature is available if the **Accurate coil voltage calculation** functionality is enabled in the **Stabilization** section of the parent feature. This functionality performs a current filtering to improve the accuracy of the computed electric field and coil voltage in time dependent and frequency domain studies. For this filtering to work properly, special conditions must be defined on the inlet/outlet boundaries, where the wire direction has a nonzero normal component. For **Linear** and **Circular** coils, the geometry analysis algorithm detects automatically these boundaries, but the **Override Coil Inlet/Outlet** feature can be used in case the detection fails. For **Numeric** coils, or when the **Conductor model** is **Single conductor**, the subnodes of the **Geometry Analysis** feature are used to identify the inlets/outlets. For **User defined** coils, the **User Defined Coil Geometry** can be used to select the inlets/outlets instead.

🔍	See Accurate Coil Voltage Calculation in the modeling section to learn more about this node.

Geometry Analysis

In 3D components, when **Conductor model** of the parent feature is **Homogenized multiturn** and **Numeric** is selected as **Coil type**, or if **Conductor model** of the parent feature

is **Single conductor** in the Coil node, the **Geometry Analysis** subnode is added by default. To this, an Input (for Geometry Analysis) subnode is also added by default.

This subnode is needed to set up the automatic analysis of the coil geometry that can determine the direction of the wires (and of the current flow). The inlet and outlet boundaries for the wires are specified using the Input (for Geometry Analysis) and Output (for Geometry Analysis) subnodes.

To complete the setup for the computation of the coil direction, a **Coil Geometry Analysis** study step must be added to the study before the main study step.

If the model represents only a part of a geometry obtained from symmetry cuts (for example, a quarter of a circle), use the settings under **Symmetry specification** to specify appropriate correction factors. Enter the **Coil length multiplication factor** F_L and **Coil area multiplication factor** F_A (dimensionless integer numbers). The actual length of the coil, used to compute the coil voltage and resistance, is then computed as the product $F_L \cdot L$, where L is the length of the domain. The cross-section area of the coil is computed as $F_A \cdot A$, where A is the cross-section area of the domain. The values of L and A are computed automatically during the Coil Geometry Analysis study step.

	• Using Coils in 3D Models
	• Automatic Coil Geometry Analysis
🔍	• The Magnetic Fields Interface
	• Theory for the Magnetic Fields Interface
	• Studies and Solvers in the *COMSOL Multiphysics Reference Manual*

Input (for Geometry Analysis)

The **Input** subnode is added by default as subnode of the Geometry Analysis node. Use it to specify the boundaries where the wires enter the domain or, in the case of a closed-loop coil, an interior boundary crossed by the wires. Used in combination with the Output (for Geometry Analysis) node, it also defines the direction of the current flow in an open coil (from **Input** to **Output**).

When applied on one or more boundaries, a red arrow in the Graphics window will show the direction of the wire. Select the **Reverse direction** checkbox to reverse the direction of the wires.

SLANTED CUT

When using the default settings, the wire direction is forced to be orthogonal to the boundary. In some cases, however, the natural direction of the current (or the wires) is at an angle with respect to the input or output normal. A typical example is a helical coil, where the input and output boundaries represent periodicity planes (see Figure 5-3).
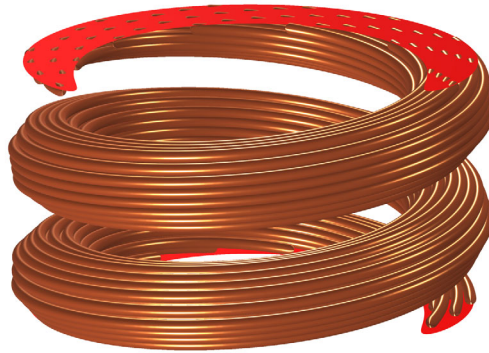
Figure 5-3: A helical coil with slanted ends. The slanted cut represents a periodicity plane.

To accommodate this case, the **Slanted cut** checkbox allows you to relax the constraints used for determining the coil geometry (it applies the boundary constraint "on average" rather than locally).

For this reason, this setting should not be used when you want to excite multiple separate coils *having different shapes* using the same input feature. If you have multiple coils with a slanted cut connected in parallel, the recommended approach is to use a separate coil feature for each one of them, and connect the coils in parallel using a circuit (or something equivalent to a circuit).

> ! For an open coil, the **Input** subnode should never be added to boundaries that are interior to the coil's domain selection. Note also that an open coil should have its start (**Input**) and end (**Output**) on boundaries that are exterior to the domain selection of the parent physics (for example, **Magnetic Fields**) and of a type that supports closure of the coil by means of an induced surface current density. The latter condition means **Magnetic Insulation** and, in the frequency domain, **Impedance Boundary Condition**.

Output (for Geometry Analysis)

The **Output** subnode is available from the context menu (right-click the Geometry Analysis parent node) or from the **Physics** toolbar, **Attributes** menu. Use it to specify the boundaries where the wires exit the domain in an open coil. Used in combination with the Input (for Geometry Analysis) node, it defines the direction of the current flow (from **Input** to **Output**). No **Output** subnode is required in a closed-loop coil.

When using the default settings, the wire direction is forced to be orthogonal to the boundary. Use the **Slanted cut** checkbox to allow for an angle between the wire direction and the output normal (see section Slanted Cut).

!	For an open coil, the **Input** subnode should never be added to boundaries that are interior to the coil's domain selection. Note also that an open coil should have its start (**Input**) and end (**Output**) on boundaries that are exterior to the domain selection of the parent physics (for example, **Magnetic Fields**) and of a type that supports closure of the coil by means of an induced surface current density. The latter condition means **Magnetic Insulation** and, in the frequency domain, **Impedance Boundary Condition**.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

🔍	• Using Coils in 3D Models • Automatic Coil Geometry Analysis • The Magnetic Fields Interface • Theory for the Magnetic Fields Interface • Studies and Solvers in the *COMSOL Multiphysics Reference Manual*

Electric Insulation (for Geometry Analysis)

The **Electric Insulation** subnode is available from the context menu (right-click the Geometry Analysis parent node) or from the **Physics** toolbar, **Attributes** menu. It is particularly useful for interior boundaries (when the coil is compacted, and the turns

touch, see Figure 5-4). It applies to the coil direction field generated by the **Coil Geometry Analysis** study step.
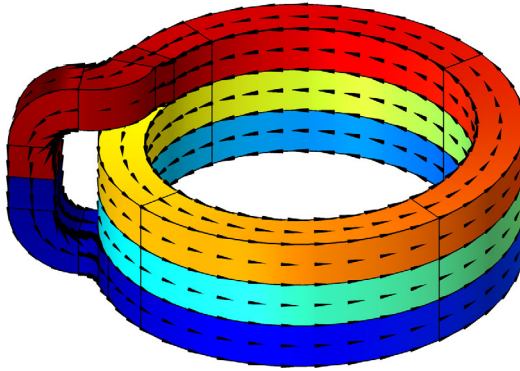
Figure 5-4: A coil geometry with turns that are geometrically unified, yet electrically separated. The electric insulation boundary condition can be used to override the electrical continuity that applies to interior boundaries by default.

⚠️
- When the **Conductor model** of the Coil feature is set to **Single conductor**, the **Electric Insulation** affects the externally applied currents only, not the currents resulting from electromagnetic induction in the model itself. *For this reason, the single conductor with electric insulation should be used in the (quasi) static regime only.*

- When the **Conductor model** of the Coil feature is set to **Homogenized multiturn**, the **Electric Insulation** works for higher frequencies as well, as both external and induced currents are forced along the same path.

- For low frequencies, the **Homogenized multiturn** option can be considered as an approximation of the **Single conductor** one. For higher frequencies, you should consider having a gap between the turns, or using the Contact Impedance boundary feature available in The Magnetic and Electric Fields Interface.

Connected Boundaries

The **Connected Boundaries** subnode is available from the context menu (right-click the Geometry Analysis parent node) or from the **Physics** toolbar, **Attributes** menu. It can

be used to mark two or more boundaries as being electrically connected for the purpose of the Coil Geometry Analysis preprocessing step.

The vector field computed in the Coil Geometry Analysis step (which corresponds to the current flow or the wire direction, according to the Conductor model selected) will have zero total flux into the selected boundaries. For example, if two nonconnected boundaries are selected, the feature will enforce the condition that the current entering one boundary must be equal to the current leaving the other boundary.

The feature models an electrical connection between the selected boundaries, and can therefore be used for coils constituted of multiple series-connected domains.

!	The **Connected Boundaries** functionality enforces current continuity as part of the Coil Geometry Analysis that computes the normalized DC current density. For Frequency Domain and Time Dependent studies, the induced current density is not subject to any restrictions by this functionality. Instead, it is recommended to set up the model with one coil per wire segment and enforce current continuity by connecting to The Electrical Circuit Interface. An alternative for Frequency Domain studies is to use The Magnetic and Electric Fields Interface and assign connected boundaries to the same Floating Potential subfeature of Magnetic Insulation.

Periodic Boundaries (for Geometry Analysis)

The **Periodic Boundaries** for the *Coil Geometry Analysis* allows you to specify on what boundaries a periodic constraint should be applied for the coil geometry analysis. The *Periodic Boundaries* feature is used together with the *Periodic Condition* feature under the physics node.

Single Conductor Coil

The **Single Conductor Coil** node is available both as a domain and as a boundary feature. It models a conductive (for example, metallic) domain or boundary subject to a lumped excitation, such as a total voltage or current. The excitation specified is translated into a conduction current flowing in the domain (as a current density) or the boundary (as a surface current density). The **Single Conductor Coil** *domain* node is a contributing feature, it is applied on top of an Ampère's Law feature that provides

the material model (electric conductivity and relative permittivity). The *boundary* node is instead exclusive, since the material model is provided in its **Thin Layer** section.

In 2D and 2D axisymmetric components, the Single Conductor Coil feature can be turned into a **Coil group** by selecting the corresponding checkbox in the **Settings** window.

> ⚠ The **Single Conductor Coil** feature is obsolete in the **Magnetic Field** and **Rotating Machinery, Magnetic** features and will be removed in future versions. The recommended alternative is to use a Coil feature with the **Conductor model** set to **Single conductor**.

> 🔍 See Modeling Coils in the modeling section to learn more about using this node.

This node is not available in 3D for The Magnetic and Electric Fields Interface and The Induction Heating Interface. In 2D and 2D axisymmetric components, 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 components is different than for 2D and 2D axisymmetric components. 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 components add boundary conditions for the coil potential. The Gap Feed (at the domain level only), Boundary Feed, Ground, and Floating Potential subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. Use these to define the geometry of the coil.

A typical setup 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. The **Gap Feed** subnode is not available for the boundary node for topological reasons.

The 3D Single Conductor Coil is applicable only in Stationary and Frequency Domain studies, since it is a local application of the physical model used in the Magnetic and Electric Fields interface.

THIN LAYER (2D AND 2D AXISYMMETRIC COMPONENTS)

This section is available for 2D and 2D axisymmetric components at the boundary level. Specify here the properties of the material constituting the thin conductive layer.

Select a **Coil conductivity** σ_{coil} (SI unit: S/m) — **From material** (the default) or **User defined**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the coil conductivity, and then enter values or expressions in the field or matrix. The default is 1 S/m.

Select a **Coil relative permittivity** $\varepsilon_{r,coil}$ (dimensionless) — **From material** (the default) or **User defined**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the coil relative permittivity, and then enter values or expressions in the field or matrix. The default is 1.

Enter the **Thickness** d_s (SI unit: m) of the thin conductive layer. The default is 1 mm.

3D Components

For 3D components, the parameters **Coil conductivity** and **Coil relative permittivity** define the material model to be used with the current continuity equation. Normally, these parameters should be the same used in the Ampère's Law feature active in the domain.

Select a **Coil conductivity** σ_{coil} (SI unit: S/m) — **From material** or **User defined**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the coil conductivity, and then enter values or expressions in the field or matrix.

Select a **Coil relative permittivity** $\varepsilon_{\text{r,coil}}$ (dimensionless) — **From material** or **User defined**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the coil relative permittivity, and then enter values or expressions in the field or matrix.

When defining this node on a boundary, also enter a **Thickness** d_{s} (SI unit: m). The default is 1 mm.

2D and 2D Axisymmetric Components

In 2D and 2D axisymmetric components, the direction of the applied electric field is assumed to be out-of-plane. In these space dimensions, the Coil group checkbox is available. Select the checkbox to activate the Coil group functionality.

The settings specifying coil name and excitation are the similar to the ones for the Coil node. See the section Coil for a description of these settings.

Gap Feed

The **Gap Feed** subnode is available from the context menu (right-click the Single Conductor Coil parent node) or from the **Physics** toolbar, **Attributes** menu for 3D components (at the domain level). **Gap Feed** is used to excite the coil from an interior boundary. It applies a discontinuity in the coil potential across a boundary to enforce a potential difference or a prescribed current. This feature must be applied to an interior boundary in the coil domain, and is typically used to excite a closed loop.

SINGLE CONDUCTOR COIL

See Single Conductor Coil for all settings.

	This node should be used with caution in frequency-domain studies. See Using Coils in 3D Models in the modeling section to learn more about this node.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

Boundary Feed

The **Boundary Feed** subnode is available from the context menu (right-click the Single Conductor Coil parent node) or from the **Physics** toolbar, **Attributes** menu for 3D components. The **Boundary Feed** node excites the coil acting on the coil potential variable. The feature must be applied to an external boundary.

SINGLE CONDUCTOR COIL

See Single Conductor Coil for all settings.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

	See Using Coils in 3D Models in the modeling section to learn more about this node.

Ground

The **Ground** subnode is available from the context menu (right-click the Single Conductor Coil parent node) or from the **Physics** toolbar, **Attributes** menu for 3D components. The **Ground** subnode enforces the condition $V_{\text{coil}} = 0$ on a boundary.

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

See Using Coils in 3D Models in the modeling section to learn more about this node.

Floating Potential

The **Floating Potential** subnode is available from the context menu (right-click the Single Conductor Coil parent node) or from the **Physics** toolbar, **Attributes** menu. It can be used to model a boundary at a constant, unknown, coil potential. If applied on multiple nonconnected boundaries, it models an electrical connection between them. The feature can be used for coils constituted of multiple series-connected domains. Unlike the corresponding feature in other AC/DC physics, this feature only allows zero total current flowing in the boundary. This is to ensure that all currents in the coils are excited by the **Boundary Feed** or **Gap Feeds** and are properly accounted for in the postprocessing variables.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

Domain Group/Boundary Group

This feature is available as a subnode of a 2D Coil in which the **Coil group** functionality has been activated. This feature is available both for domain and boundary coil features. The **Domain Group/Boundary Group** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

The Domain Group/Boundary Group feature is used to group together a number of domains/boundaries for the purposes of the coil set up. The selected domains/boundaries will be considered as one by the coil. Normally, when the **Coil group**

checkbox is selected, each domain/boundary in the parent's selection is considered as series-connected to all the other domains/boundaries.

🔍	Coil Groups

Reversed Current Direction

This feature is available as a subnode of a Coil or Single Conductor Coil in which the **Coil group** functionality has been activated. This feature is available both for domain and boundary coil features. The **Reversed Current Direction** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

Use this subnode to mark domains or boundaries in which the coil current flows in a reversed direction, that is antiparallel to the out-of-plane direction (z or azimuthal direction). This subnode can be used in any combination with the Domain Group/ Boundary Group subnode.

🔍	Coil Groups

Loss Calculation

This feature is available as a subnode to the Ampère's Law, domain Coil, Faraday's Law, Magnetic Flux Conservation, Magnet, Passive Conductor and Multiphase Winding features. This feature is used to calculate the copper loss due to the resistive heating or the iron loss due to the Hysteresis, eddy current and other effects. The feature is available with both Time Dependent and Frequency Domain studies. For Time Dependent studies, the loss calculation has to be used in combination with the Time-to-Frequency Losses study step. The Time-to-Frequency Losses is typically added after a Time Dependent study. Note that the **Loss Calculation** subnode needs to be added before the Time Dependent study is computed.

LOSS MODEL

Select a model to compute the cycle averaged loss density Q ($\mathrm{W/m^3}$). The default loss model is computed directly from the resistive heating. If the **Loss Calculation** subnode

is added to specific features and the **Constitutive Relation B-H** is set to appropriate relations, additional empirical models such as the Steinmetz and Bertotti models are available. The specific features include the Ampère's Law, the Faraday's Law and the Magnetic Flux Conservation but not the Coil. The appropriate relations include the B-H curve, the Effective B-H curve and the Hysteresis Jiles–Atherton model. The expressions of the loss density Q for different loss models are introduced as follows.

From resistive heating

$$Q = \frac{1}{T}\int_{T_e-T}^{T_e} \mathbf{J} \cdot \mathbf{E} dt$$

where T_e, T, $\mathbf{J}$, and $\mathbf{E}$ are the end time, electrical period $(1/f)$, current density and electric field, respectively.

Steinmetz

$$Q = k_h \left(\frac{f}{1[Hz]}\right)^\alpha \left(\frac{|\mathbf{B}|}{1[T]}\right)^\beta$$

where f and $\mathbf{B}$ are the frequency and magnetic flux density, respectively; k_h, α, and β are coefficients.

Bertotti

$$Q = k_h \left(\frac{f}{1[Hz]}\right)\left(\frac{|\mathbf{B}|}{1[T]}\right)^\beta + k_c \sigma f^2 |\mathbf{B}|^2 + 8\sqrt{\sigma G S V_0}\left(\frac{f}{1[Hz]}\right)^{1,5}\left(\frac{|\mathbf{B}|}{1[T]}\right)^{1,5}$$

where σ is the bulk conductivity of the laminated material; G is a constant with a value of 0.1356; S denotes the typical magnetic circuit section area; V_0 is the excess losses fitting coefficient; and k_c is the classical losses term coefficient. k_c can be computed from the lamina thickness d with the relation $k_c = \pi^2 d^2/6$ or be defined by the user. Note that the **Bertotti** model uses the material properties such as the bulk conductivity set in the interface, instead of the material data from the **Material** node.

User defined

This is an advanced setting where the loss power density as a function of the frequency (freq) and magnetic flux density (normfftB) is defined by the user.

	Generator in 2D: Application Library path **ACDC_Module/ Devices,_Motors_and_Generators/generator_2d**
	Modeling of an Electric Generator in 3D: Application Library path **ACDC_Module/Devices,_Motors_and_Generators/sector_generator_3d**
	Permanent Magnet Motor in 3D: Application Library path **ACDC_Module/ Devices,_Motors_and_Generators/pm_motor_3d**
	E-Core Transformer: Application Library path **ACDC_Module/ Devices,_Inductive/ecore_transformer**

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 in frequency-domain studies when the distance between the metallic boundaries is much smaller than the wavelength of the electromagnetic radiation. Subject to the same requirements it can also be used in the time domain as a convenient way to excite a model.

	• See About Lumped Ports and Lumped Port Parameters for more information.
	• In the time domain, use the **Magnetic and Electric Fields** interface when the excitation has DC components or when the current conservation needs to be solved (for instance in modeling conducting fluids).

An operation named as **Split by Connectivity** ( ) is available by right-clicking the parent node. The operation is useful when there are several lumped port boundaries that are not geometrically connected. Clicking the operation node will automatically generate several **Lumped Port** nodes where each selection is geometrically connected.

The geometry of the port is specified by the **Type of lumped 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 $\mathbf{a}_\mathrm{h}$ and the dimensions of the port: h_port (**Height of lumped port**) and w_port (**Width of lumped port**), both with SI unit: m.

	For 2D components, 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.

	For 2D axisymmetric components, the radial direction is in-plane with respect to the geometry. Therefore, the **Coaxial** 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.

	For 2D and 2D axisymmetric components, 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. For this case the **Direction between lumped port terminals** is assumed to be in the out-of-plane direction and the input field is not available.

LUMPED PORT PROPERTIES

Enter a unique **Lumped 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 Lumped Port

The geometry of the port is specified by the **Type of lumped 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.

🟦	Select a **Type of lumped port** — **Uniform**, **Reference-edge controlled**, **Coaxial**, **User defined**, or **Via**.

Ω	The **Type of lumped 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**. • For **Out-of-plane vector potential** the **Type of lumped port** is **User defined**. • For **In-plane vector potential** or **Three-component vector potential** select a **Type of lumped port** — **Uniform** or **User defined**.

⌒	The **Type of lumped 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**. • For **Out-of-plane vector potential** the **Type of lumped port** is **User defined**. • For **In-plane vector potential** or **Three-component vector potential** select a **Type of lumped port** — **Coaxial** or **User defined**.

Select **Reference-edge controlled** to define the **Direction between lumped port terminals** $\mathbf{a}_h$. The selected edge represents the voltage drop line.

Select **User defined** for nonuniform 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)
- A three-dimensional vector specifying the **Direction between lumped port terminals** $\mathbf{a}_h$ (the direction of the applied electric field)

Select **Via** for excitation or termination of a cylindrical shape of a structure used in a metalized via that is a plated through hole.

Terminal Type
For all **Types of lumped ports**, select a **Terminal type** — **Cable** for a voltage driven transmission line, **Current** for a current driven port, or **Circuit**.

📝	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 (connected to a passive transmission line). For **On** enter a **Voltage** V_0 (SI unit: V), and **Port phase** θ (SI unit: rad) for the **Voltage** source type or enter a **Power** P_0 (SI unit: W), that is the average input power to the lumped port, for the **Power** source type.
	The section is deactivated if a Port sweep is active, since in this case the sweep controls the ports.

💡	For the purpose of computing S-parameters, only one port at the time should be excited, for example by performing a Port sweep. In other cases, for example, when studying microwave heating, more than one inport might be desired, but in this case the S-parameter variables cannot be correctly computed. If several ports are excited at the same time, the S-parameter variables are not available for postprocessing.

SETTINGS

📝	This section is available if **Cable** or **Current** is selected as the **Terminal type**.
	• For **Cable** enter the **Characteristic impedance** Z_{ref} (SI unit: Ω).
	• For **Current** enter a **current** I_0 (SI unit: A).

Lumped Element

Use a **Lumped Element** node to mimic the insertion of a resistor, capacitor, inductor, or general impedance between two metallic boundaries. It is similar to the Lumped Port but does not allow for an excitation and does not generate S-parameter output.

LUMPED ELEMENT PROPERTIES
Enter a unique **Lumped element name**. See Lumped Port for the rest of the settings.

Select a **Lumped element type** — **User defined** (the default), **Inductor**, or **Capacitor**.

- For **User defined** enter a **Lumped element impedance** Z_{element} (SI unit: Ω; default 50 Ω.)

- For **Inductor** enter a **Lumped element inductance** L_{element} (SI unit: H; default 1 nH).

- For **Capacitor** enter a **Lumped element capacitance** C_{element} (SI unit: F; default 1 pF).

Edge Current

Use the **Edge Current** node to specify a line current along one or more edges. A positive edge current flows in the direction marked by the red arrow.

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or click **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

EDGE CURRENT

Specify the **Edge current** I_0 (SI unit: A). The default is 0 A.

External Magnetic Vector Potential

	This node 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 physics 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.

Impedance Boundary Condition

	This node is available with the Frequency Domain study type.

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 conductive material excluded from the model.

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.
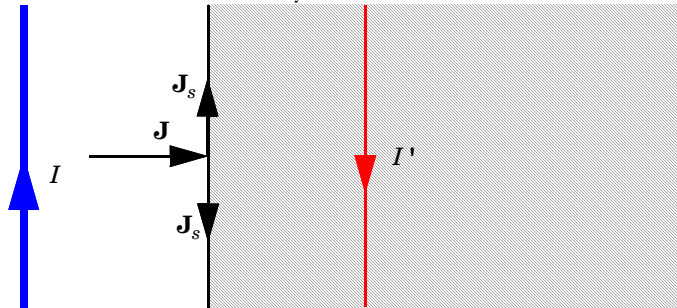
The impedance boundary condition is used on exterior boundaries representing the surface of a lossy domain. The shaded (lossy) region is not part of the model. The effective induced image currents are of reduced magnitude due to losses. Any current flowing into the boundary is perfectly balanced by induced surface currents as for the perfect electric conductor boundary condition. The tangential electric field is generally small but nonzero at the boundary.

The **Harmonic Perturbation** subnode (it is of the exclusive type) is available from the context menu (right-click the parent node) or in the **Physics** toolbar, click the **Attributes** menu and select **Harmonic Perturbation**. For more information see Harmonic

Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

Use it to specify a perturbation source electric field.

IMPEDANCE BOUNDARY CONDITION

The following material properties can be defined for the domain outside the boundary, which this boundary condition approximates. The default use the values **From material**. For **User defined** enter different values or expressions.

- **Relative permittivity**, ε_r (dimensionless)
- **Relative permeability**, μ_r (dimensionless)
- **Electric 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

📝   This node is available with the Frequency Domain study type.

The **Transition Boundary Condition** is used on interior boundaries to model a sheet of a highly conducting medium with a thickness relatively smaller than the characteristic size, and curvature, of the objects being modeled. It can be used even if the thickness is many times greater than the skin depth. Accounting for the material properties as well as the thickness of the sheet, transfer and surface impedances are used to relate the current flowing on the surface of either side to the 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. See Ref. 7 for more details.

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**, taking the properties from the material specified for the boundary. For **User defined**, enter different values or expressions.

- **Relative permittivity**, ε_r (dimensionless)
- **Relative permeability**, μ_r (dimensionless)
- **Electric conductivity**, σ (SI unit: S/m)
- **Surface thickness**, d_s (Si unit: m)

Layered Transition Boundary Condition

The **Layered Transition Boundary Condition** is an extension of the **Transition Boundary Condition** that allows to model a sequence of thin metallic layers using a **Layered Material**. It represents a discontinuity in the tangential electric field. For each layer in the **Layered Material**, the transfer and surface impedances are obtained from the layer thickness and material properties. The impedances are then used to relate the discontinuity in the tangential electric field to the current flowing on the surface of either side (up/down) of the corresponding layer. Mathematically this reads:

$$\mathbf{J}_{s,\,\text{up},\,i} = \frac{(Z_S\mathbf{E}_{t,\,i} - Z_T\mathbf{E}_{t,\,i-1})}{Z_S^2 - Z_T^2}$$

$$\mathbf{J}_{s,\,\text{down},\,i} = \frac{(Z_S\mathbf{E}_{t,\,i-1} - Z_T\mathbf{E}_{t,\,i})}{Z_S^2 - Z_T^2}$$

$$Z_S = \frac{-j\omega\mu}{k}\frac{1}{\tan(kd)}$$

$$Z_T = \frac{-j\omega\mu}{k}\frac{1}{\sin(kd)}$$

$$k = \omega\sqrt{(\varepsilon + (\sigma/(j\omega)))\mu}$$

where the index $i = 1, 2, …, n$ refers to the layer number. The system of equations above is solved for each layer in the **Layered Material**. The index i has been omitted from the expressions of the impedances and the wave vector k in order to improve their readability.
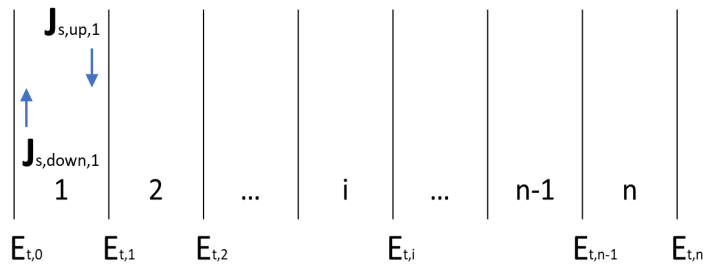
Figure 5-5: The layered material is composed of n layers. The surface currents on the up and downside of each layer are determined from the transfer and surface impedances and are functions of the tangential electric fields.

- For more information, see Layer and Interface Selection Tools.
- See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the *COMSOL Multiphysics Reference Manual* for details on the definition of layered materials.

SHELL PROPERTIES

The **Shell Properties** section displays which layered material(s) the **Layered Transition Boundary Condition** is coupled to.

Clear the **Use all layers** checkbox in order to select a specific layered material from the list. The **Layered Transition Boundary Condition** feature is then applicable only on the boundaries where the chosen material is defined.

You can visualize the selected layered material and the layers that constitute it by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

The thickness of the layered material should be set as follows, depending on the type of material:

- In a **Material** node, the layer **Thickness** is set in the **Material Contents** section by adding a **Shell** property group from the **Material Properties** section in the material **Settings** window. This automatically adds a **Shell** subnode under the **Material** node, transforming it into a layered material.

- When the layered material is a **Single Layer Material**, the **Thickness** is set in the **Material Contents** section in the **Settings** window. Alternatively it can be set in the **Layer Definition** section of the **Shell** property group **Settings** window.

- For a general **Layered Material**, added through a **Layered Material Link** or a **Layered Material Stack**, the **Thickness** is set in the **Layer Definition** section of the **Settings** window. Several layers may be defined in the table, and the **Thickness** should be defined for each of them. The total thickness of the layered material is the sum of all the layers thicknesses.

LAYERED TRANSITION BOUNDARY CONDITION

The **Layered Transition Boundary Condition** section lists the material properties of the thin layered material that this boundary condition approximates:

- **Relative permittivity**, ε_r (dimensionless)
- **Relative permeability**, μ_r (dimensionless)
- **Electric conductivity**, σ (SI unit: S/m)

The defaults use the values **From material**, taking the properties from the layered material specified for the boundary. Otherwise, choose **User defined** and enter different values or expressions. In the latter case all layers constituting the layered material will take on the same value for the selected property.

Layered Impedance Boundary Condition

The **Layered Impedance Boundary Condition** is an extension of the **Impedance Boundary Condition** that allows to model a sequence of geometrically thin layers on top of a

substrate. It is used on exterior boundaries where the field is known to penetrate only a short distance outside the boundary. In brief, this feature combines a **Layered Transition Boundary Condition** with an **Impedance Boundary Condition** of the type:

$$\sqrt{\frac{\mu_0 \mu_r}{\varepsilon_c}} \mathbf{n} \times \mathbf{H} + \mathbf{E} - (\mathbf{n} \cdot \mathbf{E})\mathbf{n} = 0 .$$

The layer stack is built using a **Layered Material**. For each layer in the **Layered Material**, the transfer and surface impedances are obtained from the layer thickness and the material properties. The impedances are then used to relate the discontinuity in the tangential electric field to the current flowing on the surface of either side (up/down) of the corresponding layer. The mathematical details relative to the field propagation in the layer stack can be found in the Layered Transition Boundary Condition section.

🔍	• See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the *COMSOL Multiphysics Reference Manual* for details on the definition of layered materials.

📝	The **Layered Impedance Boundary Condition** is based on the assumption that in the thin layers and in the substrate the wave propagates essentially in the normal direction. Thus, the wave could be incident in the normal direction or the wave could be refracted to propagate in a direction close to the normal direction. The latter condition is fulfilled for a good conductor.
	The thickness of the layer should be less than the radius of curvature for the boundary.

SHELL PROPERTIES

The **Shell Properties** section displays which **Layered Material** is coupled to the **Layered Impedance Boundary Condition**.

Clear the **Use all layers** checkbox in order to select a specific **Layered Material** from the list. The **Layered Impedance Boundary Condition** feature is then applicable only on the boundaries where the chosen **Layered Material** is defined.

You can visualize the selected **Layered Material** and the layers that constitute it by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

The thickness of the **Layered Material** should be set as follows, depending on the type of material:

- For a general **Layered Material**, added through a **Layered Material Link** or a **Layered Material Stack**, the **Thickness** is set in the **Layer Definition** section of the **Settings** window. Several layers may be defined in the table, and the **Thickness** should be defined for each of them. The total thickness of the **Layered Material** is the sum of all the layers thicknesses.

- When the **Layered Material** is a **Single Layer Material**, the **Thickness** is set in the **Material Contents** section in the **Settings** window. Alternatively it can be set in the **Layer Definition** section of the **Shell** property group **Settings** window.

- In a **Material** node, the layer **Thickness** is set in the **Material Contents** section by adding a **Shell** property group from the **Material Properties** section in the material **Settings** window. This automatically adds a **Shell** subnode under the **Material** node, transforming it into a **Layered Material**.

LAYER PROPERTIES

The **Layer Properties** section specifies the material properties of the thin layers constituting the stack located on top of the substrate. The defaults use the values **From material**. In this case, the material properties are taken layer by layer from the **Layered Material** existing on those boundaries where the **Layered Impedance Boundary Condition** feature is enabled. Otherwise, choose **User defined** and enter a value or an expression. In the latter case all layers constituting the chosen **Layered Material** will take on the same given value for the selected material property.

SUBSTRATE PROPERTIES

The **Substrate Properties** section specifies the material properties of the thick domain that is not included in the model, being approximated by an **Impedance Boundary Condition**. Select a **Substrate Material** from the list of materials that have been introduced in the model previously. The defaults use the values **From material**. In this case, the material properties are taken from the specified **Substrate Material**. Otherwise, choose **User defined** and enter a value or an expression.

Surface Current Density (Subnode)

The **Surface Current Density** subnode is available from the context menu (right-click the Transition Boundary Condition parent node) or in the **Physics** toolbar, click the

Attributes menu. It adds a surface current density on upside and downside of the transition boundary.

SURFACE CURRENT DENSITY

Enter values or expressions for the **Surface Current Density** on upside $\mathbf{J}_{s0u}$ and downside $\mathbf{J}_{s0d}$ (SI unit: A/m) coordinates. The defaults are 0 A/m.

Thin Low Permeability Gap

Use the **Thin Low Permeability Gap** boundary condition to model gaps in magnetic materials filled with another material with smaller relative permeability. This feature applies the condition

$$\mathbf{n} \times (\mathbf{H}_1 - \mathbf{H}_2) = \nabla_t \times \frac{d}{\mu_0 \mu_r} \nabla_t \times \mathbf{A}$$

This boundary condition is only applicable on interior boundaries and pair boundaries. Use this boundary condition to model, for example, air gaps in magnetic circuits. When used in the **Magnetic and Electric Fields** interface, the subnodes Electric Insulation, Contact Impedance, and Electrical Contact can be added as electrical boundary conditions.

THIN LOW PERMEABILITY GAP

The default **Relative permeability** μ_r (dimensionless) is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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. The default is 1 m.

Magnetic Point Dipole

Apply a **Magnetic Point Dipole M** (SI unit: m^2·A) 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 finite. The dipole moment is a vector entity with the positive direction set by the curl of the current. Magnetic point dipoles are available with 2D and 3D components. See Magnetic Point Dipole (on Axis) for the 2D axisymmetric settings.

DIPOLE SPECIFICATION

Select a **Dipole specification** — **Magnitude and direction** or **Dipole moment**.

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: m²·A).

If **Dipole moment** is selected under **Dipole Specification**, enter coordinates for the **Magnetic dipole moment** $\mathbf{m}$ (SI unit: m²·A).

Magnetic Point Dipole (on Axis)

Use the **Magnetic Point Dipole (on Axis)** to apply a point dipole to points on a 2D axisymmetric component. See Magnetic Point Dipole for settings for 2D and 3D components.

MAGNETIC POINT DIPOLE (ON AXIS)

Enter a **Magnetic dipole moment in z direction** m_z (SI unit: m²·A).

Magnetic Shielding

The **Magnetic Shielding** node adds a boundary condition for magnetic shielding. It describes a thin layer of a highly permeable medium that shields the magnetic field. Because of the presence of this thin layer, the tangential components of the magnetic field and vector potential have discontinuities given by:

$$\mathbf{n} \times (\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{n} \times \mathbf{H}_t(\mathbf{B}_t)$$

$$\mathbf{B}_t = \mathbf{n} \times \left(\frac{\mathbf{A}_1 - \mathbf{A}_2}{d_s}\right)$$

In this equation, d_s is the surface layer thickness and the relation between the tangential magnetic field $\mathbf{H}_t$ and flux density $\mathbf{B}_t$ inside the layer can be linear and given by a relative permeability or nonlinear and given by the B-H curve of the layer material.

> 📝 The Magnetic Shielding model uses a magnetostatic approach and can only be used in a **Stationary** study step. For a frequency-domain model of a similar physical object (thin layer) use the Transition Boundary Condition node instead.

MAGNETIC SHIELDING

Enter a value or expression for the **Surface thickness** d_s (SI unit: m). Select a **Magnetization model**, either **Relative permeability** (the default), or **B-H curve**.

Relative Permeability

Select **Relative permeability** μ_r (dimensionless) to use the constitutive relation $\mathbf{B} = \mu_0\mu_r\mathbf{H}$. The default uses values **From material**. For **User defined**, enter another value or expression.

B-H Curve

Select **B-H curve** to use a curve that relates the magnetic flux density $\mathbf{B}$ and the magnetic field $\mathbf{H}$ as $|\mathbf{B}| = f(|\mathbf{H}|)$. The **Magnetic field norm** setting can take the values **From material** or **User defined**.

Reference

1. David C. Meeker, "An improved continuum skin and proximity effect model for hexagonally packed wires," *Journal of Computational and Applied Mathematics*, vol. 236, no. 18, pp. 4635-4644, 2012.

The Magnetic and Electric Fields Interface

The **Magnetic and Electric Fields (mef)** interface ( ), found under the **AC/DC > Electromagnetic Fields > Vector Formulations** branch when adding a physics interface, is used to compute magnetic field and current distributions when the exciting current is driven by an applied voltage. Stationary, time-dependent, and frequency-domain modeling are supported in 2D and 3D. Note that in most cases, using the Magnetic Fields interface with its dedicated coil modeling features is the preferred choice over using the Magnetic and Electric Fields interface.

The physics interface solves Maxwell's equations formulated using the magnetic vector potential and the scalar electric potential as the dependent variables.

> In three-dimensional components, 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 nonconductive, it is recommended to use the Magnetic Fields interface and apply the Single Turn Coil domain node on the conductive regions, instead of using this physics interface.

Use the Magnetic and Electric Fields interface when the current continuity equation is needed everywhere in the simulation domain.

The main node 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 electric conductivity.

The Ampère's Law and Current Conservation feature can be overridden with an **Ampère's Law** feature or a **Coil** feature, that are identical to the ones in the **Magnetic Fields** interface, removing the electric potential and the current conservation equation from the selected domains.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Free Space**, **Magnetic Insulation** (the default boundary condition for the magnetic vector potential), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and external currents. You

can also right-click **Magnetic and Electric Fields** to select physics features from the context menu.

Physics-Controlled Mesh

The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the **Sequence type** is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the checkbox in the **Use** column on the same table row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

Information from the physics, such as the presence of an infinite elements domain or periodic condition, will be used to automatically set up an appropriate meshing sequence.

!	In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh.

📄	Except where described below, most **Settings** windows are the same as for the Magnetic Fields, Electrostatics, and Electric Currents interfaces. Also see About the Magnetic and Electric Fields Interface Boundary Conditions for more information.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is mef.

BACKGROUND FIELD

The only option available from the **Solve for** list is **Full field**.

For 2D and 2D axisymmetric components, 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).

In 2D axisymmetric dimension, The Covariant Formulation is used.

THICKNESS

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 components.

INPUT SWEEP SETTINGS

Enter a **Reference impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω. This impedance is used by **Terminal** features (when the **Terminal Type** is set to **Terminated**) and the **Touchstone file export** functionality.

Select the **Activate input sweep** checkbox to switch on the sweep and invoke a parametric sweep over the Lumped ports or the Terminal nodes.

Select an option from the **Sweep on** list — **Terminals** or **Ports**. This setting controls which features are activated during the sweep.

Enter a **Sweep parameter name** to indicate the name of the model parameter that controls the terminal or port activated in each step of the sweep. The default is PortName. The name given must match the model parameter, defined under **Global Definitions**, that is the object of a **Parametric Sweep** node in the current **Study**.

The lumped parameters computed can be subject to a **Touchstone file export**. To activate this functionality, enter a file path or **Browse** for a file. Select a **Parameter format (value pairs)** for the Touchstone export — **Magnitude and angle (MA)** (the default), **Magnitude in dB and angle (DB)**, or **Real and imaginary parts (RI)**. Select an option from the **If file exists** list — **Overwrite** or **Create new**. Create new is useful when the model is

solved multiple times with different settings. Select a **Parameter to export** — **Z** (the default), **Y**, or **S**.

When **Terminals** is selected under **Sweep on**, also select a **Parameter to export** — **Z** (the default), **Y**, or **S**. When the sweep is carried on **Ports**, the exported parameter is always **S**.

ERROR CHECK

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

When the **Check applicability of features in study** checkbox is selected, any features that are incompatible with the study will generate an error message when trying to solve or show the default solver. No solver will be generated. Deselect it and you will be able to run the model, possibly with runtime errors instead. It is available to allow the advanced user to tweak any feature and use it outside of its intended study scope.

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

Select the shape order for the **Magnetic vector potential** dependent variable — **Linear**, **Quadratic** (the default), or **Cubic** and for the **Electric potential** dependent variable — **Linear**, **Quadratic** (the default), **Cubic**, or **Quartic**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

🔍	• About the Magnetic and Electric Fields Interface Boundary Conditions • Computing Lumped Parameters • Domain, Boundary, Edge, Point, and Pair Nodes for the Magnetic and Electric Fields Interface • Theory for the Magnetic and Electric Fields Interface

- *Magnetic Brake*: Application Library path **ACDC_Module/ Devices,_Transducers_and_Actuators/magnetic_brake**

- *Inductance of a Power Inductor*: Application Library path **ACDC_Module/Devices,_Inductive/power_inductor**

About the Magnetic and Electric Fields Interface 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 can lead to unphysical models that violate the current conservation law.

The basic steps for this are to add the magnetic boundary conditions from the **Physics** toolbar. Then right-click these magnetic boundary condition nodes to add compatible electric boundary conditions as subnodes.

STEP 1: MAGNETIC BOUNDARY CONDITIONS

With no surface currents present the physics 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 **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

Depending on specific modeling problems, different electric boundary conditions should be used. To include the modeling of the full-wave effect, for instance, in the frequency domain studies, the most general exterior electric boundary condition is the **Ground**; that is,

$$V = 0$$

In some other cases where conducting current is more significant, other electric boundary conditions can be used. For example, at exterior boundaries between

conducting and nonconducting materials, the general boundary condition is the **Electric Insulation**, expressed as

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

which is a natural boundary condition.

> A default **Ground** subnode is also added under the default **Magnetic Insulation** node. This is to ensure that default boundary conditions are always specified both for the magnetic vector potential and the electric potential. Additional subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

Table 5-3 lists the interior and exterior boundary conditions available with this physics interface.

TABLE 5-3: INTERIOR AND EXTERIOR ELECTRIC AND MAGNETIC BOUNDARY CONDITIONS FOR THE MAGNETIC AND ELECTRIC FIELDS INTERFACE.

NODE	INTERIOR	EXTERIOR
MAGNETIC BOUNDARIES		
Background Magnetic Flux Density		x
Impedance Boundary Condition		x
Lumped Port	x	x
Magnetic Continuity	x	
Magnetic Field		x
Magnetic Insulation	x	x
Magnetic Potential	x	x
Magnetic Shielding	x	
Perfect Magnetic Conductor	x	x
Surface Current	x	x
Thin Low Permeability Gap	x	
Transition Boundary Condition	x	
Coil	x	
ELECTRIC BOUNDARIES		
Contact Impedance	x	
Electric Insulation	x	x
Electric Potential	x	x

TABLE 5-3: INTERIOR AND EXTERIOR ELECTRIC AND MAGNETIC BOUNDARY CONDITIONS FOR THE
MAGNETIC AND ELECTRIC FIELDS INTERFACE.

NODE	INTERIOR	EXTERIOR
Electric Shielding	x	
Floating Potential	x	x
Ground	x	x
Normal Current Density		x
Periodic Condition		x

BOUNDARY CONDITION COMBINATIONS

Table 5-4 shows the possible combination of magnetic and electric boundary
conditions, with an explanation of the physical model represented by the more
advanced combinations.

TABLE 5-4: POSSIBLE COMBINATIONS OF ELECTRIC AND MAGNETIC BOUNDARY CONDITIONS FOR THE
MAGNETIC AND ELECTRIC FIELDS INTERFACE.

MAGNETIC FEATURE	ELECTRIC FEATURE	PHYSICAL MODEL
Magnetic Insulation	Electric Insulation	
	Electric Potential	
	Ground	
	Floating Potential	
	Terminal	
	Normal Current Density	
Magnetic Continuity	<none>	<no effect>
	Electric Insulation	Thin nonconducting, nonmagnetic layer, such as an air gap or crack in a metal. It can be used to model internal insulation.
	Contact Impedance	Thin resistive, nonmagnetic layer, such as an oxide layer in a metal.
Magnetic Shielding	<none>	Magnetic (high-μ_r) layer with same conductive properties as the surrounding medium.
	Electric Shielding	Thin magnetic, highly conductive layer (high-μ_r, high-σ).
	Electric Insulation	Thin nonconducting magnetic layer.
	Contact Impedance	Thin resistive magnetic layer.
Surface Current	<none>	Externally imposed surface current density.

MAGNETIC FEATURE	ELECTRIC FEATURE	PHYSICAL MODEL
	Electric Shielding	Thin nonmagnetic, highly conductive layer along which a surface current density flows, that is automatically included in the parent node.
Background Magnetic Flux Density	Electric Insulation	
	Electric Potential	
	Ground	
	Floating Potential	
	Terminal	
	Normal Current Density	

Domain, Boundary, Edge, Point, and Pair Nodes for the Magnetic and Electric Fields Interface

The Magnetic and Electric Fields Interface has these domain, boundary, edge, point, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

> In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

Also see About the Magnetic and Electric Fields Interface Boundary Conditions and Table 5-3 for a list of interior and exterior boundary conditions.

> For axisymmetric components, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an **Axial Symmetry** node to the component that is valid on the axial symmetry boundaries only.

These nodes are described in this section:

- Free Space
- Ampère's Law and Current Conservation
- Background Magnetic Flux Density
- Initial Values
- Magnetic Continuity

- Magnetic Insulation
- Magnetic Shielding
- RLC Coil Group
- Surface Current

The following nodes are described for the Magnetic Fields or Electric Currents interfaces,

- Ampère's Law
- Override Coil Inlet/Outlet
- Contact Impedance
- Edge Current
- Electric Insulation
- Electric Point Dipole
- Electric Point Dipole (on Axis)
- Electric Potential
- Electric Shielding
- External Current Density
- Floating Potential
- Force Calculation
- Gauge Fixing for A-Field
- Ground
- Impedance Boundary Condition

- Lumped Element
- Lumped Port
- Magnetic Field
- Magnetic Point Dipole
- Magnetic Potential
- Coil
- Normal Current Density
- Perfect Magnetic Conductor
- Periodic Condition
- Single Conductor Coil
- Terminal
- Thin Low Permeability Gap
- Transition Boundary Condition
- Velocity (Lorentz Term)

🔍	- Sector Symmetry - Infinite Elements, Perfectly Matched Layers, and Absorbing Layers in the *COMSOL Multiphysics Reference Manual*

> ! In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Free Space

For inductive and capacitive (sometimes resistive) analysis, it is quite common for the fields to extend into the environment of the device in question. It is therefore important not only to model the device itself, but also its (close) surroundings. Typically, this is done by adding a box or sphere around the device's geometry.

The default **Free Space** node is used to specify the physical conditions in close proximity to the device — *typically in air or vacuum*. It adds the Ampère's law and current conservation *governing equations* with a limited set of material models: the permeability μ_r and permittivity ε_r are assumed to be one, and the conductivity is taken from the **Stabilization conductivity**.

The Free Space feature provides a canvas on top of which other features can be added to locally specify material properties and excitation forms.

STABILIZATION

Stabilization Conductivity
The **Stabilization conductivity** allows for four options:

- **From material**; use the conductivity specified by the material.
- **Automatic** (default); sets a conductivity $\sigma_{stab} = 1 \, \text{S/m}$ for stationary and time dependent studies, and $\sigma_{stab} = 0$ otherwise. This is done in order to avoid getting a singular problem, which can happen when the conductivity is zero. For more information, see Theory for the Magnetic and Electric Fields Interface.
- **Off**; disable the stabilization conductivity altogether: $\sigma_{stab} = 0$. This models a perfect vacuum and is a good approximation for air.
- **User defined**; specify a suitable stabilization conductivity directly.

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. There are two types of this feature available, **Ampère's Law and Current Conservation in Solids** and **Ampère's Law and Current Conservation in Fluids**. This distinction decides how materials behave and how material properties are interpreted when the mesh is deformed.

Ampère's Law and Current Conservation in Solids applies to materials whose properties change as functions of material strain, material orientation, and other variables evaluated in a material reference configuration (material frame).

Ampère's Law and Current Conservation in Fluids applies to materials whose properties are defined only as functions of the current local state at each point in the spatial frame, and for which no unique material reference configuration can be defined.

!	In COMSOL versions **6.2** and earlier, this **Solids**/**Fluids** distinction was controlled within the **Ampère's Law and Current Conservation** node by specifying **Solid**/**Nonsolid** in the **Material type** setting.

📄	These features can be overridden with an Ampère's Law feature or a Coil feature, which are identical to the ones in the **Magnetic Fields** interface, removing the electric potential and the current conservation equation from the selected domains.

📄	The **Constitutive Relation Jc-E** and **Constitutive Relation B-H** settings are the same as Ampère's Law for the Magnetic Fields interface. The **Constitutive Relation D-E** settings are the same as for Current Conservation for the Electric Currents interface.

Initial Values

The **Initial Values** node 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.

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 Wb/m and 0 V, respectively.

Magnetic Insulation

The **Magnetic Insulation** node is the default magnetic boundary condition for the Magnetic and Electric Fields interface and sets the tangential components of the magnetic potential to zero at the boundary $\mathbf{n} \times \mathbf{A} = 0$. Electric boundary conditions can be applied by adding subnodes to the **Magnetic Insulation** node. The default electric boundary condition, added under each **Magnetic Insulation** node, is **Ground**. The **Electric Potential**, **Electric Insulation**, **Floating Potential**, **Terminal**, or **Normal Current Density** subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

⊚	About the Magnetic and Electric Fields Interface Boundary Conditions

Surface Current

The **Surface Current** node adds a boundary condition for a surface current density $\mathbf{J}_s$.

An externally applied surface current density can be specified directly in the **Settings** window for the node. Alternatively, the **Electric Shielding** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. Use the subnode to model a thin conductive layer (a layer whose electric conductivity s is higher than in the surrounding domains). In this case, the electric current flows preferably along the conductive boundary, and the resulting surface current density is automatically incorporated in the parent **Surface Current** node to act as a source for the magnetic field.

SURFACE CURRENT

Enter values or expressions for the **Surface current density $\mathbf{J}_{s0}$** (SI unit: A/m) coordinates.

Magnetic Shielding

The **Magnetic Shielding** node adds a boundary condition for magnetic shielding. By itself, this feature represents a geometrically thin layer with a relative permeability substantially higher than the surrounding material, that shields the magnetic field.

Subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. Use the subnodes to provide an electrical model

on top of the Shielding magnetic model. The available subnodes are Electric Shielding, Electric Insulation, Contact Impedance, and Electrical Contact. The possible combinations are:

- **Magnetic Shielding** with no subnode: this represents a highly magnetic layer with no special electrical properties.

- **Magnetic Shielding** and **Electric Shielding**: this combination represents a layer in which both the relative permeability μ_r and the electric conductivity σ are much higher than in the adjacent domain. This layer shields the magnetic field and the current preferentially flows along it. The resulting surface current density is incorporated in the model as a source for the magnetic field.

- **Magnetic Shielding** and **Electric Insulation**: this combination represents a nonconductive magnetic layer. The layer shields the magnetic field and there is no current flowing through it. The electric potential is discontinuous across the boundary.

- **Magnetic Shielding** and **Contact Impedance** or **Electrical Contact**: this combination represents a resistive magnetic layer. The layer shields the magnetic field and the current flows preferably orthogonally to the layer. The electric potential is discontinuous across the boundary.

MAGNETIC SHIELDING

The default **Relative permeability** μ_r (dimensionless) uses values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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).

Magnetic Continuity

The **Magnetic Continuity** node does not apply any condition or constraint to the model. Its purpose is to allow the application of electrical boundary conditions by means of the subnodes Electric Insulation, Contact Impedance, and Electrical Contact. The subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. Use this feature to model thin layer with particular electrical properties but with the same magnetic properties of the surrounding medium.

RLC Coil Group

	This node is available for 2D and 2D axisymmetric components. This node also can only be applied on domains where an Ampère's Law feature is active.

The **RLC** (Resistive-Inductive-Capacitive) **Coil Group** node is an advanced coil feature that takes advantage of the A-V formulation of the Magnetic and Electric Fields features to take into account in-plane electric current flow in the coil's current balance. This feature combines the magnetic model of a **Single Turn Coil** (in Coil group mode) with a multiple-terminal electrical model for the in-plane current. This node can be used to approximate in 2D a coil in which (due to capacitive coupling or other phenomena) there is relatively significant current flow in the in-plane directions.

Apply this feature to a group of domains representing the cross sections of the coil turns on the modeling plane. Since the electric potential is assumed to be constant in the cross section of each turn, the V variable must be removed in the selected domains by applying an **Ampère's Law** node. The feature applies an external current density flowing orthogonally to the plane, and also imposes a voltage constraint on the boundary of each cross section, computed from the coil and the excitation properties. The feature also enforces a balance of the current flowing out-of-plane and the current leaking in the plane between the coil turns.

	The approximation applied by this feature is valid only under the assumption that the current flowing in-plane is relatively small compared with the current in the coil. If this condition is not fulfilled, the model cannot be considered two-dimensional and a full 3D component is required.

DOMAINS LATTICE

In order to apply the correct voltage at each coil turn, it is necessary to precisely define the order in which the domains are connected. The RLC Coil Group feature assumes that all domains are of the same shape and are disposed on a two-dimensional *Bravais lattice*; that is, that there exist two vectors **a** and **b** (called *primitive vectors*) such that given two lattice points $\mathbf{r}_1$ and $\mathbf{r}_2$ the following relation holds:

$$\mathbf{r}_1 - \mathbf{r}_2 = n\mathbf{a} + m\mathbf{b}$$

with n and m being integers. One-dimensional lattices (linear lattices) are also supported by this feature.

The RLC Coil Group node can automatically determine two lattice vectors (or one, in the case of a linear lattice) from the geometry and position of the selected domains. The choices **Row-wise** and **Column-wise** in the **Ordering** settings use this functionality. The two choices control how the vectors **a** and **b** are chosen.

Since the primitive vectors of a Bravais lattice are not unique, the lattice recognition algorithm could find two vectors different from the desired one. In this case, it is possible to manually specify **User defined** lattice vectors.

DOMAIN ORDERING

By default, the RLC Coil Group feature assigns the first turn index (closest to the reference potential) to the turn with the smallest n and m. When using the automatic recognition algorithm, this usually corresponds to the coil turn with smallest x and y (or r and z in axisymmetry). The other coil turns are numbered in order of increasing m first, and then increasing n. Refer to the diagram in the node's **Settings** window for a visual explanation.

If **Winding** is set to **Alternated**, the direction of increasing m is reversed each time n changes value. Again, refer to the diagram for a visual explanation.

RESISTIVE-INDUCTIVE-CAPACITIVE COIL MODEL

The RLC Coil Group feature expands the model applied by a Single Conductor Coil domain feature by including a current balance for each turn and constraints for the voltage. The feature introduces $n + 1$ voltages (where n is the number of coil turns) with the interpretation of electric potential values at the turn end terminals: V_0, V_1,..., V_n, where V_0 is the value specified as the **Ground voltage** in the **Settings** window. The cross section shown in the interface corresponds to the midsection of the turns, that is half way between the end terminals. The i-th coil turn ($i = 1,..., n$) has an applied potential difference

$$V_i^d \ = \ V_i - V_{i-1}$$

that drives an out-of-plane external current density computed as in the Single Conductor Coil case:

$$\mathbf{J}_e = \sigma \frac{V_i^d}{d} \mathbf{e}_z \qquad \text{in 2D}$$

$$\mathbf{J}_e = \sigma \frac{V_i^d}{2\pi r} \mathbf{e}_\phi \qquad \text{in 2D axisym.}$$

On the boundaries of the turn domain the electric potential variable V is constrained to the value $(V_i + V_{i-1})/2$, that is the mean of the electric potential values of its end terminals.

The current flowing in-plane is balanced with the difference of the currents between two adjacent turns. If I_i is the out-of-plane current flowing in the ith turn,

$$I_i = \int_{\Omega_i} (\mathbf{J} \cdot \mathbf{e})$$

then the current balance for the coil turns is

$$I_i - \int_{\partial\Omega_i} (\mathbf{J} \cdot \mathbf{n}) - I_{i+1} = 0 \qquad i = 1, ..., n-1$$

$$I_n - \int_{\partial\Omega_n} (\mathbf{J} \cdot \mathbf{n}) - I_{\text{Coil}} = 0$$

The coil current I_{Coil} is computed from the coil excitation in the same way as for the Single Conductor Coil feature.

RLC COIL GROUP

The settings in this section is very similar to the ones described in the Single Conductor Coil node. The only addition is the **Ground voltage** input. Specify in the text box the voltage at one end of the coil (SI unit: V). The voltage at the other end depends on the excitation type. This setting is useful, for example, to connect in series two coil groups.

GEOMETRY

This section specifies the information needed to identify the Bravais lattice on which the coil turns are placed The feature automatically detects the two primitive vectors of the lattice if **Ordering** is set as **Row-wise** (the default) or **Column-wise**. For **User defined** use the diagram as a guide to manually enter the **Primitive vectors a** and **b** (SI unit: m) of the lattice. The diagram shows how the domains are ordered for each choice.

Select a **Winding** style — **Regular** (the default) or **Alternated**. Look at the diagram for an illustration of these options.

Background Magnetic Flux Density

The **Background Magnetic Flux Density** node adds a boundary condition for the magnetic vector potential:

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

where $\mathbf{A}_b$ is defined to satisfy:

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

where $\mathbf{B}_b$ is defined in the **Magnetic Flux Density** section.

As the equation suggests, the boundary condition only guarantees that the tangential components of $\mathbf{A}$ and $\mathbf{A}_b$ are equal. Their normal components might deviate depending on specific situations. The **Background Magnetic Flux Density** node is incompatible with the **Periodic Condition** node. If periodicity is required in a background field, use the **Magnetic Potential** node along with the **Periodic Condition** node, and ensure that appropriate values are set consistently.

MAGNETIC FLUX DENSITY

Enter a value or expression for the **Magnetic flux density** $\mathbf{B}_b$ (SI unit: T) components.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

The Magnetic Field Formulation Interface

The **Magnetic Field Formulation (mfh)** interface (🧲), found under the **AC/DC >
Electromagnetic Fields** > **Vector Formulations** branch when adding a physics interface, is
used to compute magnetic fields and induced current distributions. It is especially
suitable for time-domain magnetic modeling of materials with a strongly nonlinear E-J
characteristic such as superconductors. Stationary, frequency-domain, small-signal
analysis and time-domain modeling are supported in 2D and 3D. Note that the
formulation requires a nonzero conductivity everywhere.

The physics interface solves Maxwell's equations formulated using the magnetic field
as the dependent variable.

The main node is Faraday's Law, which adds the equation for the magnetic field and
provides an interface for defining the constitutive relations and its associated properties
such as the relative permeability.

When this physics interface is added, these default nodes are also added to the **Model
Builder** — **Magnetic Field Formulation**, **Faraday's Law**, **Magnetic Insulation** (the default
boundary condition), **Initial Values**, and **Magnetic Gauss' Law**. Then, from the **Physics**
toolbar, add other nodes that implement, for example, boundary conditions and
external currents. You can also right-click **Magnetic Field Formulation** to select physics
features from the context menu.

Physics-Controlled Mesh
The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the
Sequence type is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled
Mesh** section, find the physics interface in the **Contributor** column and select or clear
the checkbox in the **Use** column on the same table row for enabling (the default) or
disabling contributions from the physics interface to the physics-controlled mesh.

Information from the physics, such as the presence of an infinite elements domain, will
be used to automatically set up an appropriate meshing sequence.

!	In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh.

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is mfh.

COMPONENTS

This section is only available in 2D and 2D axially symmetric components.

The electric currents and the electric field vectors are orthogonal to the magnetic field. 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 in-plane components only, allowing an out-of-plane current flow.

Select **Components** — **Out-of-plane magnetic field**, **In-plane magnetic field** (the default), or **Three-component magnetic field** for the magnetic field. This choice affect also the directions in which the electric current is allowed to flow: the magnetic field and electric currents are orthogonal.

When **In-plane magnetic field** or **Three-component magnetic field** is selected, the Magnetic Gauss' Law node becomes available, and is added as default.

DEPENDENT VARIABLES

The dependent variable is the **Magnetic field H**. You can change both its field name and the individual component variable names. If the new field name coincides with the name of another magnetic vector potential field in the model, the interfaces share degrees of freedom and component names. The new field name must not coincide with the name of a field of another type or with a component name belonging to some other field. Component names must be unique within a model, except for fields of the same type sharing a common field name.

DIVERGENCE CONSTRAINT

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

The **Activate divergence constraint** checkbox is selected by default, which enables the default Magnetic Gauss' Law node. Click to clear the checkbox to disable the Magnetic Gauss's Law node and remove it from the physics interface.

ERROR CHECK

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

When the **Check applicability of features in study** checkbox is selected, any features that are incompatible with the study will generate an error message when trying to solve or show the default solver. No solver will be generated. Deselect it and you will be able to run the model, possibly with runtime errors instead. It is available to allow the advanced user to tweak any feature and use it outside of its intended study scope.

DISCRETIZATION

Select the shape order for the **Magnetic field** dependent variable — **Linear**, **Linear type 2**, **Quadratic** (the default), **Quadratic type 2**, **Cubic**, or **Cubic type 2**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

🔍	• Domain, Boundary, Point, and Pair Nodes for the Magnetic Field Formulation Interface • Theory for the Magnetic Field Formulation Interface

---

▥	*Superconducting Wire*: Application Library path **ACDC_Module/ Devices,_Superconducting/superconducting_wire**

Domain, Boundary, Point, and Pair Nodes for the Magnetic Field Formulation Interface

The Magnetic Field Formulation Interface has these domain, boundary, point, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

💡	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

About the Boundary Conditions

With no surface currents present the physics interface conditions

$$\mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{0}$$

need to be fulfilled. Because $\mathbf{H}$ is being solved for, the tangential component of the magnetic field is always continuous, and thus the condition is automatically fulfilled.

Table 5-5 lists the interior and exterior boundary conditions available with this physics interface.

TABLE 5-5: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS FOR THE MAGNETIC FIELD FORMULATION INTERFACE.

NODE	INTERIOR	EXTERIOR
Electric Field	x	x
Impedance Boundary Condition		x
Magnetic Field	x	x
Magnetic Insulation	x	x
Perfect Magnetic Conductor	x	x
Surface Magnetic Current Density	x	x

Available Nodes

These nodes are available for this physics interface and listed in alphabetical order. Also see Table 5-5 for the list of interior and exterior boundary conditions.

- Electric Field
- Faraday's Law
- Initial Values

- Magnetic Gauss' Law
- Magnetic Insulation
- Magnetic Field
- Surface Magnetic Current Density

These nodes are described for the Magnetic Fields or Electrostatics interface (listed in alphabetical order):

- External Current Density
- Force Calculation
- Impedance Boundary Condition
- Perfect Magnetic Conductor
- Velocity (Lorentz Term)

For 2D axisymmetric components, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an **Axial Symmetry** node to the component that is valid on the axial symmetry boundaries only.

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Faraday's Law

The **Faraday's Law** node adds Faraday's law for the electric field and provides an interface for defining the constitutive relation and its associated properties such as the relative permeability as well as electric properties.

Except for two extra options; **Resistivity** and **E-J characteristic**, in the **Constitutive Relation Jc-E** settings, and the important notes listed below, see Ampère's Law for the rest of the settings.

If **E-J characteristic** is selected in the **Conduction Current** settings, the External Current Density node cannot be used to drive currents in that domain selection.

In order to obtain a closed system of equations with this physics interface, the **Electric conductivity** (defined under **Conduction Current**) must be nonzero in all domains.

Initial Values

The **Initial Values** node adds an initial value for the magnetic field **H** that can serve as an initial value for a transient simulation or as an initial guess for a nonlinear solver.

INITIAL VALUES

Enter values or expressions for the **Magnetic field H** (SI unit: A/m) components. The defaults are 0 A/m.

Magnetic Gauss' Law

The **Magnetic Gauss' Law** node is used to impose the condition

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

on the magnetic flux density by introducing an auxiliary dependent variable ψ with the corresponding equation.

MAGNETIC GAUSS' LAW

The variable ψ is used to impose a condition on the derivatives of the magnetic flux density, so its absolute value does not have particular significance; only its gradient enters the equations (the variable ψ acts more or less like a potential). The absolute value of the variable can be set by entering the Enter a value or expression for the **Divergence condition variable scaling** Ψ_0 (SI unit: A). The default is 1 A, which is appropriate in most cases.

ADVANCED SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

This section allows a more fine control on the boundary conditions for ψ applied by the **Magnetic Gauss' Law** feature. The domain equation for ψ only imposes a condition on the gradient, so it is important to constrain the absolute value of ψ to ensure a nonsingular model. Select the **Constrain variable in at least one point** to ensure that there is always a constraint set on the value of the divergence condition variable ψ. The feature also sets up automatically the appropriate boundary conditions for ψ. Use the **Method** list to select the approach used to enforce the boundary condition: **Constrain value** (the default) or **Constrain tangential gradient**.

Magnetic Insulation

The **Magnetic Insulation** node is the default boundary condition for the Magnetic Field Formulation interface. In this formulation, this condition corresponds to setting the tangential components of the electric field to zero at the boundary $\mathbf{n} \times \mathbf{E} = 0$.

	For more information and the settings, see Magnetic Insulation as described for the Magnetic Fields interface.

Electric Field

The **Electric Field** boundary condition

$$\mathbf{n} \times \mathbf{E} = \mathbf{n} \times \mathbf{E}_0$$

specifies the tangential component of the electric field. This feature can be used to specify a source electric field on the boundary of the conductor.

The **Harmonic Perturbation** subnode (it is of the exclusive type) is available from the context menu (right-click the parent node) or in the **Physics** toolbar, click the **Attributes** menu and select **Harmonic Perturbation**. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

ELECTRIC FIELD

Enter the value or expression for the components of the **Electric field $\mathbf{E}_0$** (SI unit: V/m). The defaults are 0 V/m.

Surface Magnetic Current Density

The **Surface Magnetic Current Density** node adds a boundary condition for a surface magnetic current density $\mathbf{J}_{ms}$ (a flow of magnetic charges along a boundary):

$$-\mathbf{n} \times \mathbf{E} = \mathbf{J}_{ms}$$
$$\mathbf{n} \times (\mathbf{E}_1 - \mathbf{E}_2) = \mathbf{J}_{ms}$$

These expressions apply to exterior and interior boundaries, respectively. On exterior boundaries, the condition is equivalent to an Electric field boundary condition where the electric field is specified as $\mathbf{J}_{ms} \times \mathbf{n}$.

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or click **Harmonic Perturbation** in the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

SURFACE MAGNETIC CURRENT DENSITY

Enter values or expressions for the **Surface magnetic current density $\mathbf{J}_{ms,\,0}$** (SI unit V/m) coordinates. The defaults are 0 V/m.

Magnetic Field

The **Magnetic Field** boundary condition

$$\mathbf{n} \times \mathbf{H} = \mathbf{n} \times \mathbf{H}_0$$

specifies the tangential component of the magnetic field. As the equation suggests, the boundary condition only guarantees that the tangential components of $\mathbf{H}$ and $\mathbf{H}_0$ are equal. Their normal components might deviate depending on specific situations.

The **Harmonic Perturbation** subnode (it is of the exclusive type) is available from the context menu (right-click the parent node) or in the **Physics** toolbar, click the **Attributes** menu and select **Harmonic Perturbation**. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

MAGNETIC FIELD

Enter the value or expression for the components of the **Magnetic field $\mathbf{H}_0$** (SI unit: A/m). The defaults are 0 A/m.

Magnetic Fields Formulation Magnetic Fields No Currents Coupling

The **Magnetic Fields Formulation Magnetic Fields No Currents Coupling** multiphysics node assures continuity of the normal magnetic flux density and the tangential magnetic field across boundaries between The Magnetic Fields, No Currents Interface and The Magnetic Field Formulation Interface. The **Magnetic Fields Formulation Magnetic Fields No Currents Coupling** is available from the Multiphysics menu in the Physics toolbar or by right-clicking the Multiphysics Couplings node in Model Builder, if both The Magnetic Fields, No Currents Interface and The Magnetic Field Formulation Interface are available.

BOUNDARY SELECTION

Select Manual or All boundaries from the Selection list. Make additional edits to the list of boundary entity numbers using the Selection toolbar buttons. When All boundaries is selected from the Selection list, the boundaries exterior to The Magnetic Fields, No Currents Interface that intersect the exterior boundaries to The Magnetic Field Formulation Interface are available in the boundary entity number list.

COUPLED INTERFACES

Select The Magnetic Fields, No Currents Interface as Primary interface and The Magnetic Field Formulation Interface as Secondary interface.

The Magnetic Fields, Currents Only Interface

The **Magnetic Fields, Currents Only (mfco)** interface (⊘), found under the **AC/DC** > **Magnetic Fields** > **Vector Formulations** branch when adding a physics interface, is used to compute magnetic fields from currents under the assumption that all regions have a uniform relative magnetic permeability of one. It is designed to support both divergence-free and non-divergence-free currents. In free space it returns the value of the Biot–Savart integral. The interface supports the *Stationary*, *Stationary Source Sweep*, and *Frequency Domain Source Sweep* study steps in 3D.

The physics interface solves Ampère's law using the magnetic vector potential as the dependent variable. The main node is the **Free Space** feature which solves Ampère's law in free space. It is designed to efficiently compute the lumped inductance matrix of complex circuits. Due to the support of modeling non-divergence-free currents, this interface is able to calculate the partial contributions of magnetic fields generated from open conductors.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Magnetic Fields, Currents Only**, **Free Space, Exterior Boundaries** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, additional boundary conditions and point conditions. You can also right-click **Magnetic Fields, Currents Only** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `mfco`.

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

Select the shape order for the **Magnetic vector potential** dependent variable — **Linear**, **Quadratic** (the default), **Cubic**, or **Quartic**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

🔍	• Domain, Boundary, Point, and Pair Nodes for the Magnetic Fields, Currents Only Interface
	• Theory for the Magnetic Fields, Currents Only Interface

▦	• *Magnetic Field of a Helmholtz Coil*: Application Library path **ACDC_Module/Devices,_Inductive/helmholtz_coil**
	• *Inductance Matrix Calculation of PCB Coils*: Application Library path **ACDC_Module/Devices,_Inductive/inductance_matrix_pcb_coils**

Domain, Boundary, Point, and Pair Nodes for the Magnetic Fields, Currents Only Interface

The Magnetic Fields, Currents Only Interface has these domain, boundary, point, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

💡	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

These nodes are available for this physics interface and listed in alphabetical order. Also see Table 5-6 for a list of interior and exterior boundary conditions.

- Conductor
- Electric Insulation (Subnode)[1]
- Exterior Boundaries
- External Current Density
- Free Space
- Ground (Subnode)[1]

- Initial Values
- Magnetic Continuity
- Magnetic Insulation[1]
- Magnetic Potential
- Passive Conductor
- Terminal (Subnode)

[1] Described for another physics interface in this user guide.

Table 5-6 lists the interior and exterior boundary conditions available with this physics interface.

TABLE 5-6: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS FOR THE MAGNETIC FIELDS, CURRENTS ONLY INTERFACE.

PHYSICS NODE	INTERIOR	EXTERIOR
Exterior Boundaries		x
Magnetic Potential		x
Magnetic Insulation		x
Terminal (Subnode)	x	x
Ground (Subnode)	x	x
Electric Insulation (Subnode)	x	x

Free Space

The **Free Space** node is the default domain feature of the Magnetic Fields, Currents Only interface. It adds the Ampère's law in free space and assumes that all domains have a uniform relative magnetic permeability of one. Also see Theory for the Magnetic Fields, Currents Only Interface.

Exterior Boundaries

The **Exterior Boundaries** node is the default boundary condition of the Magnetic Fields, Currents Only interface. It is usually used with the Stationary Source Sweep with

Initialization study step. The **Exterior Boundaries** adds a constraint over the boundaries of the model to the value of an equivalent source based on a monopole and dipole expansion of the physical sources, expressed as

$$\mathbf{A} = \frac{\mu_0}{4\pi}\left(\frac{\mathbf{Q}_m}{|\mathbf{r}|} + \frac{(\mathbf{D}_m - (\mathbf{Q}_m \otimes \mathbf{r})) \cdot \mathbf{r}}{|\mathbf{r}|^3}\right)$$

where $\mathbf{r} = \mathbf{x} - \mathbf{x}_s$, and $\mathbf{x}_s$ is the point that an equivalent point source is placed. $\mathbf{Q}_m$ and $\mathbf{D}_m$ are the source contributions from the magnetic monopole and dipole, respective. They are computed as integrals over current sources expressed as

$$\mathbf{Q}_m = \int \mathbf{J}$$

$$\mathbf{D}_m = \int (\mathbf{J} \otimes \mathbf{r})$$

> [!]
> The **Exterior Boundaries** feature is designed to efficiently compute the lumped inductance matrix with the Stationary Source Sweep with Initialization study step. For other studies such as the Stationary study, it is computationally heavier. In these cases, to improve the efficiency, switch the **Exterior Field Equivalent Source** to **User defined** or use the Magnetic Insulation boundary condition to override it. One can also use the **Infinite Element** feature to remove the constraint over adjacent **Exterior Boundaries**.

Initial Values

The **Initial Values** node adds an initial value for the magnetic vector potential.

INITIAL VALUES

Enter a value or expression for the initial value of the **Magnetic vector potential A** (SI unit: Wb/m).

Conductor

The **Conductor** node adds the continuity equation for the electric potential and provides an interface for defining the electric conductivity. Use it as an easy way to

apply current sources, especially in connection with the Stationary Source Sweep with Initialization study step. The Conductor feature has several subnodes (**Terminal**, **Ground**, **Electric Insulation**) that can be used to define the boundary conditions for the current conservation.

> [!] Instead of using the Conductor node to compute the current flow field, one can also use the External Current Density feature. The current can be either defined by an expression or imported from other interfaces such as The Electric Currents Interface.

An operation named as **Split by Connectivity** (✎) is available by right-clicking the node. This operation is useful when there are several conductor domains that are not geometrically connected. Clicking the operation node will automatically generate several Conductor nodes where each selection is geometrically connected.

Multi-Terminal Conductor

The **Multi-Terminal Conductor** node adds the continuity equation for the electric potential and provides an interface for defining the electric conductivity. Use it as an easy way to apply current sources, especially in connection with the Stationary Source Sweep with Initialization study step and for conductors containing multiple current inputs to sweep onto. The Multi-Terminal Conductor feature has several subnodes (**Terminal**, **Ground**, **Electric Insulation**) that can be used to define the boundary conditions for the current conservation.

An operation named as **Split by Connectivity** (✎) is available by right-clicking its Terminal subfeature node. This operation is useful when there are several terminals in the conductor domains that are not geometrically connected. Clicking the operation node will automatically generate several Terminal nodes where each selection is geometrically connected.

Terminal (Subnode)

The **Terminal** subnode provides an interface to define the total current flowing in the conductor (parent node).

External Current Density

The **External Current Density** node adds an externally generated current density $\mathbf{J}_e$, which appears on the right-hand side of the equation that The Magnetic Fields, Currents Only Interface defines.

Magnetic Potential

The **Magnetic Potential** node adds a boundary condition for the magnetic vector potential:

$$\mathbf{A} = \mathbf{A}_0$$

MAGNETIC POTENTIAL

Enter a value or expression for the **Magnetic vector potential** $\mathbf{A}_0$ (SI unit: Wb/m) coordinates.

Magnetic Continuity

The **Magnetic Continuity** node adds a continuity pair feature for the magnetic vector potential. This node is only suitable for pairs where the boundaries match.

BOUNDARY SELECTION

The selection list in this section shows the boundaries for the selected pairs.

PAIR SELECTION

From the **Pairs** list, choose **From list** (the default), **All**, **All contact pairs**, or **All identity pairs**. If you choose **From list**, select the pairs where you want to impose continuity across the pair boundaries by clicking the **Add** button (+) and choose the pairs to add from the list in the **Add** dialog that opens; then click **OK**. Also see Identity, Contact, and Sector Symmetry Pairs.

Passive Conductor

The **Passive Conductor** node adds a simplified version of the Conductor feature, which does not require specification of terminal and ground boundaries, and is not swept over by source sweep studies. It is used to model a conductor without feeding. It is still possible to add an Electric Insulation subfeature that can be used to define the boundary conditions for the current conservation.

PASSIVE CONDUCTOR

The settings in this section are identical to the ones in the Conductor feature, except for that you cannot enter a name. This is not needed in the **Passive Conductor** feature, since it is not used in source sweeps.

The Rotating Machinery, Magnetic Interface

The **Rotating Machinery, Magnetic (rmm)** interface (🕐), found under the **AC/DC >
Electromagnetics and Mechanics** branch when adding a physics interface, is used for
design and analysis of electric motors and generators. Stationary and time-domain
modeling is supported in 2D and 3D.

The physics interface solves Maxwell's equations formulated using a combination of
magnetic vector potential and magnetic scalar potential as the dependent variables. For
a theoretical discussion, see The A Formulation, The Vm Formulation, and The A–Vm
Formulation.

When this physics interface is added, these default nodes are also added to the **Model
Builder**: **Ampère's Law**, **Mixed Formulation Boundary**, **Magnetic Insulation** (the default
boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other
nodes that implement, for example, boundary conditions and point conditions. You
can also right-click **Rotating Machinery, Magnetic** to select physics features from the
context menu.

⚠	• The physics interface only works correctly if the geometry is created as an assembly pair from individual composite objects for the rotor and stator parts. An **Identity Boundary Pair** feature must also be defined for the rotor-stator physics interface boundaries under the **Model>Definitions** node in the **Model Builder**. This is usually done by choosing **Form Assembly** in the finalization node of the geometry sequence. • It is important to avoid having scalar potential regions that surround vector potential regions in which nonclosed currents can flow. For a detailed analysis, see this blog post https://www.comsol.com/blogs/how-to-model-rotating-machinery-in-3d/

Physics-Controlled Mesh
The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the
Sequence type is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled
Mesh** section, find the physics interface in the **Contributor** column and select or clear

the checkbox in the **Use** column on the same table row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

Information from the physics, such as the presence of an infinite elements domain or periodic condition, will be used to automatically set up an appropriate meshing sequence.

!	In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh.

📄	Only nodes unique to the Rotating Machinery, Magnetic interface are described in this section. Most nodes are described for The Magnetic Fields Interface or The Magnetic Fields, Currents Only Interface.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `rmm`.

BACKGROUND FIELD

Select an option from the **Solve for** list — **Full field** (the default) or **Reduced field**. For **Reduced field** specify a **Background magnetic vector potential** $\mathbf{A}_b$ (SI unit: Wb/m). The entered expressions must be differentiable.

📄	Notice that expressions including coupling operators are not differentiable and cannot be used as background fields.

The total field used in the physics and equations are given by the sum of the reduced and background fields.

COMPONENTS

For 2D components, select the **Components** — **Out-of-plane vector potential** (the default), **In-plane vector potential**, or **Three-component vector potential**.

THICKNESS

For 2D components, 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 components.

PORT SWEEP SETTINGS

Select the **Activate port sweep** checkbox and enter a **Sweep parameter name** in the field. The default is PortName.

The lumped parameters computed can be subject to a **Touchstone file export**. To activate this functionality, enter a file path or click **Browse** to navigate to a file. Select a **Parameter format (value pairs)** for the Touchstone export — **Magnitude and angle (MA)** (the default), **Magnitude in dB and angle (DB)**, or **Real and imaginary parts (RI)**. Select the desired operation to perform from the **If file exists** list — **Overwrite** (the default) or **Create new**. The latter is useful when the model is solved multiple times with different settings. Enter a **Reference impedance, Touchstone file export** Z_{ref} (SI unit: Ω). The default is 50 Ω.

ERROR CHECK

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

When the **Check applicability of features in study** checkbox is selected, any features that are incompatible with the study will generate an error message when trying to solve or show the default solver. No solver will be generated. Deselect it and you will be able to run the model, possibly with runtime errors instead. It is available to allow the advanced user to tweak any feature and use it outside of its intended study scope.

DEPENDENT VARIABLES

The dependent variables (field variables) are the *magnetic vector potential* **A** and the *magnetic scalar potential* V_{m}. The name can be changed but the names of fields and dependent variables must be unique within a model.

Select the shape order for the **Magnetic vector potential** and the **Magnetic scalar potential** dependent variables — **Linear**, **Quadratic** (the default), or **Cubic**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

🔍	Domain, Boundary, Edge, Point, and Pair Nodes for the Rotating Machinery, Magnetic Interface

▥	• *Generator in 2D*: Application Library path **ACDC_Module/ Devices,_Motors_and_Generators/generator_2d** • *Rotating Machinery 3D Tutorial*: Application Library path **ACDC_Module/Devices,_Motors_and_Generators/ rotating_machinery_3d_tutorial**

Domain, Boundary, Edge, Point, and Pair Nodes for the Rotating Machinery, Magnetic Interface

The Rotating Machinery, Magnetic Interface has these boundary, edge, pair, and point nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

💡	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

These domain nodes are described in this section:

- Initial Values
- Mixed Formulation Boundary

- Arkkio Torque Calculation
- Multiphase Winding

> [!] In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

The following nodes are described for the **Magnetic Fields** or **Magnetic Fields, No Currents** interfaces:

- Ampère's Law[1]
- Boundary Feed
- Coil[1]
- Edge Current
- Electric Point Dipole
- Electric Point Dipole (on Axis)
- External Current Density[1]
- External Magnetic Vector Potential
- Force Calculation
- Gap Feed
- Ground
- Laminated Core[3]
- Laminated Core[4]
- Line Current (Out-of-Plane)
- Lumped Element[1]
- Lumped Port[1]
- Magnet[5]
- Magnet[6]

- Magnetic Field[1]
- Magnetic Flux Conservation[2]
- Magnetic Flux Density[2]
- Magnetic Point Dipole
- Magnetic Potential[1]
- Magnetic Scalar Potential[2]
- Magnetic Shielding[1]
- Override Coil Inlet/Outlet
- Passive Conductor
- Perfect Magnetic Conductor[1]
- Periodic Condition
- Sector Symmetry
- Single Conductor Coil[1]
- Surface Current[1]
- Thin Low Permeability Gap[1]
- Velocity (Lorentz Term)[1]
- Zero Magnetic Scalar Potential[2]

[1] Select this from the **Magnetic Fields** submenu for this interface.

[2] Select this from the **Magnetic Fields, No Currents** submenu for this interface.

[3] This feature is called **Laminated Core, Ampère's Law** in this interface.

[4] This feature is called **Laminated Core, Magnetic Flux Conservation** in this interface.

[5] This feature is called **Conducting Magnet** in this interface.

[6] This feature is called **Nonconducting Magnet** in this interface.

Initial Values

The **Initial Values** node adds an initial value for the magnetic vector potential.

Enter a value or expression for the initial value of the **Magnetic vector potential A** (SI unit: Wb/m).

Magnetic Insulation

The **Magnetic Insulation** node is the default external boundary condition for the Rotating Machinery, Magnetic interface. It adds a boundary condition that, on boundaries with magnetic vector potential (Ampère's Law domains) sets the tangential components of the magnetic potential to zero at the boundary $\mathbf{n} \times \mathbf{A} = 0$. On external boundaries with magnetic scalar potential (Magnetic Flux Conservation domains) it sets the normal component of the magnetic flux density to zero $\mathbf{n} \cdot \mathbf{B} = 0$.

> As external boundary condition, it is useful at boundaries confining a surrounding region of air or to model symmetry cuts.

For the magnetic vector potential, Magnetic insulation is a special case of the magnetic potential boundary condition that sets the tangential component of the magnetic potential to zero and can be used for the modeling of a lossless metallic surface, for example, a ground plane or as a symmetry type boundary condition. The Magnetic Insulation node can also be applied on interior boundaries in vector potential domains. For more details on this, see also the corresponding documentation section in the Magnetic Fields interface on Magnetic Insulation.

Mixed Formulation Boundary

The **Mixed Formulation Boundary** node imposes continuity between the magnetic scalar potential on one side of the boundary and the magnetic vector potential on the other side. This feature is added by default and is applied automatically to all appropriate boundaries.

Arkkio Torque Calculation

Use the **Arkkio Torque Calculation** node to define globally available torque variables for the selected domains, which is the air gap. Compared with the **Force Calculation** feature, the **Arkkio Torque Calculation** requires a much coarser mesh for the same accuracy. For a theoretical discussion, see the section Computing Forces and Torques.

Enter a **Torque name**, which is then appended to global variables. This feature also gives access to the azimuthal and radial force density and magnetic flux density. The computed torque is made available as a global scalar variable with name `<name>.Tark_<torque name>`, where `<name>` is the **Name** of the physics interface.

The **Arkkio Torque Calculation** node has to be used together with **Continuity**, **Sector Symmetry**, or **Arkkio Extra Circular Boundaries** features. The **Arkkio parameters** and **Number of sectors** default to be automatically retrieved in most cases. Otherwise, **User defined** options are available.

ADVANCED SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog.

This section enables you to set a **Tolerance for automatic entity identification** when **Arkkio parameters** is set to **Automatic**.

Arkkio Extra Circular Boundaries

Use the **Arkkio Extra Circular Boundaries** node to specify extra circular boundaries that are not defined by the **Continuity** or **Sector Symmetry** for the **Arkkio Torque Calculation** feature. This could be useful when modeling electric motors without using **Assembly**.

APPLICABLE PAIR REGION

In **Allowed region** section, select **Fallback and nonpair regions**, **Nonpair region**, or **All regions** to specify different pair regions to the selected boundaries.

Multiphase Winding

In the context of electrical machines, a system of coils having different phase offsets is commonly referred to as a winding. The **Multiphase Winding** feature simplifies the excitation of a system of coils having multiple phase angles. It is available in **The Rotating Machinery**, **Magnetic Interface** (RMM), and **The Magnetic Machinery, Rotating, Time Periodic Interface** (MMTP). For the RMM interface, it is possible to add a Loss Calculation subfeature in order to compute induced losses, whereas for MMTP this is built into the main **Multiphase Winding** feature.

Winding Name

Enter a **Winding name**. This name is appended to the variables defined by this winding.

Excitation Type

Select the **Excitation type** to be used. With **Phase currents** the peak value of phase currents applied to all phases is specified directly as an input. With **DQ currents** the excitation is specified in terms of direct and quadrature current components. The combination of Park and Clarke transformations (Ref. 1) simplifies winding excitation with respect to the direct and quadrature axis of a rotating magnetic field.

Peak Phase Current

This field is only available when **Phase currents** is chosen as the **Excitation type**. Enter the **Peak phase current**, which is the peak value of the current applied to all phases.

Initial Electrical Angle

This field is only available when **Phase currents** is chosen as the **Excitation type**. Enter the **Initial electrical angle**, which is an angular offset applied to all phases of the system. Typically, the excitation in a multiphase system needs to be adjusted in order to be in a particular angular position relative to other components of the overall system.

D-axis Current

This field is only available when **DQ currents** is chosen as the **Excitation type**. Enter the **D-axis current** which is the component of the winding current producing a magnetic field along the direct axis.

Q-axis Current

This field is only available when **DQ currents** is chosen as the **Excitation type**. Enter the **Q-axis current** which is the component of the winding current producing a magnetic field along the quadrature axis.

Initial D-axis Angle

This field is only available when **DQ currents** is chosen as the **Excitation type**. Enter the **Initial D-axis angle** which aligns the reference frame for the direct-quadrature transformations with the direct axis of the rotor magnetic field. The angle is specified

as the winding electrical angle at which the concatenated flux of the reference phase (phase offset angle 0°) reaches its peak value.

> ❗ The DQ post-processing variables are always available regardless of which **Excitation type** has been selected. If **Phase Currents** is selected as **Excitation type**, it is assumed that the **Initial D-axis angle** is equal to zero. To obtain consistent DQ quantities in this case the initial rotor position should be aligned to give peak concatenated flux in the reference phase (phase offset angle 0°).

Frequency for Time-Dependent Studies

Enter the **Frequency for time-dependent studies**, which is the electrical frequency of excitation that defines the speed of a moving or rotating magnetic field.

Reference Frame Definition

This field is only available when **DQ currents** is chosen as the **Excitation type**. Select the **Reference frame definition** to be used in the direct-quadrature transformation. **Synchronous**, being the default, defines the reference frame as rotating at the same speed as the electrical excitation. For a synchronous machine this means the winding direct-quadrature variables are referred to a reference frame fixed with the rotor. **Stationary** defines the reference frame to be non-rotating in which the direct-quadrature variables are effectively the result of a pure Clarke transformation or equivalent to alpha-beta components. **User defined** allows for arbitrary definition of the **Reference frame angle**. This angle can be a function of time, making it possible to define any angular velocity for the reference frame definition which is also known as *arbitrary reference frame* (Ref. 2). For The **Magnetic Machinery, Rotating, Time Periodic Interface** the only available **Reference frame definition** is **Synchronous**.

Winding Layout Configuration

Select the **Winding layout configuration** to be used. The layout of a winding is the pattern in which the coils are arranged with regard to phase and geometrical position. The different options supported by the **Multiphase Winding** feature are **User defined** and **Automatic three phase**. When **User defined** is selected, multiple Phase subfeatures can be added, in which domains representing coils of each phase can be selected. Select **Automatic three phase** to instead configure the domains of all phases under the assumption that the winding is a balanced three phase system where all coils are of equal shape and number of turns, and that all coil positions or slots are fully utilized.

Number of Poles

This field is only available when **Automatic three phase** is chosen as the **Winding layout configuration**. Specify the **Number of poles** of the magnetic field to be excited by the winding.

Number of Slots

This field is only available when **Automatic three phase** is chosen as the **Winding layout configuration**. Specify the **Number of slots** available for all coils of the winding.

Number of Coils Per Slot

This field is only available when **Automatic three phase** is chosen as the **Winding layout configuration**. Specify the **Number of coils per slot**, which is how many coils each slot can accommodate. The automatic winding layout supports either one or two coils per slot.

Add Phases

This button is only available when **Automatic three phase** is chosen as the **Winding layout configuration**. When this button is clicked, a check will first be made to see if the number of poles, slots, and coils per slot are valid under the assumptions described for the **Automatic three phase** setting. If this check passes, three **Phase** subfeatures will be added under the **Multiphase Winding** node, with selections of coil domains and corresponding phase angles automatically configured.

> 💡 It is not guaranteed the algorithm will identify a pattern with minimal coil span for all combinations of number of slots and poles. The resulting pattern can be inspected by looking at the selection of the generated subfeatures and freely modified to correct any shortcomings.

SECTOR SETTINGS

Sector Settings

This section is only available when **Automatic three phase** is chosen as the **Winding layout configuration**. This setting allows for custom configuration of sector periodicity in models where only a part of the full geometry is included. The two different options in this section are **Automatic** and **User defined**. If **Automatic** is chosen, the settings will automatically match those in any **Continuity** or **Sector Symmetry** feature that is present in the model. When **User defined** is chosen, the options become identical to those in the Sector Symmetry feature.

HOMOGENIZED MULTITURN CONDUCTOR

This section is identical to the corresponding section in the Coil feature.

Phase

This subnode can be added under the **Multiphase Winding** node in order to select domains representing coils with the same **Phase offset angle**. If using the **Automatic three phase** winding layout is used, three copies of this feature will be generated automatically.

PHASE

Phase Offset Angle
Enter the **Phase offset angle** for this phase.

Reversed Current Direction

This feature can be added as a subnode under the **Phase** node, and is used to select domains representing parts of coils where the current direction is opposite to the normal direction (out of plane).

References for Rotating Machinery, Magnetic Interface

1. D. Hanselman, *Brushless Motors: Magnetic Design, Performance, and Control of Brushless DC and Permanent Magnet Synchronous Motors*, E-Man Press LLC, pp. 231-256, 2012

2. P. Krause, O. Wasynczuk, S. Sudhoff and S. Pekarek, *Analysis of Electric Machinery and Drive Systems*, 3rd ed., Wiley-IEEE Press, pp. 86-115, 2013.

The Magnetic Machinery, Rotating, Time Periodic Interface

The **Magnetic Machinery, Rotating, Time Periodic (mmtp)** interface ( ), found under the **AC/DC** > **Electromagnetics and Mechanics** branch when adding a physics interface, is used for design and analysis of rotating magnetic machines such as electric motors and generators. Time Periodic equation form is enforced, but Stationary, Frequency, and Time Dependent studies are allowed for coupling to other physics. The interface supports 2D and in-plane magnetic fields.

The physics interface solves Maxwell's equations formulated using out-of-plane magnetic vector potential, see The A Formulation, Time Periodic, and Time Periodic Equation.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Free Space**, **Exterior Boundaries** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, external sources and boundary conditions. You can also right-click **Magnetic Machinery, Rotating, Time Periodic** to select physics features from the context menu.

!	• The physics interface only works correctly when modeling rotating machinery if the geometry is created as an assembly pair from individual composite objects for the rotor and stator parts, and proper **Rotating Domain** and **Rotational Continuity Pair** features are added. The rules are similar to that of the **Rotating Machinery, Magnetic** interface. However, each **Rotating Domain** should also cover exactly one geometry component.
	• Rotation features can be specified only if the **Motion type** setting in the **Motion Settings** section is set to **Rotational motion**. The button **Add Rotational Boundary Features** in that section can be used to simplify the model setup for the most typical geometries.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `mmtp`.

THICKNESS

Here, it is possible to enter a value for the **Out-of-plane thickness** d (SI unit: m). The default value is 1 m and affects lumped quantities such as axial torques, forces, and voltages.

TIME PERIODIC SETTINGS

Here, it is possible to enter values for the following inputs:

- **Time periodic electrical frequency** f_{TP} (SI unit: Hz). The default value is 50 Hz, but it is possible to enter a different value or to have the frequency valued to `freq` if this variable is supplied by the study, as it is when having frequency domain study steps.

- **Number of time frames** n_{TP}, which is an even integer that must be equal to or larger than 2. This specifies the number of points that are used to discretize the time period (or in other words, the phase interval $\phi \in [0, 2\pi]$). Here, ϕ is an independent variable that can be accessed in user inputs as $\phi_{\mathrm{TP}} = \phi$ by using the expression `<phystag>.phaseTP` (for example, `mmtp.phaseTP`). For more details, see Feature Specific Variables and Accessing the Solution at Different Phases. The **Number of time frames** is also accessible as a variable named `<phystag>.nTP` (for example, `mmtp.nTP`).

- **Number of variable groups**, which is a setting that is useful for decreasing RAM consumption, as it simplifies the setup of segregated solver strategies. The **Number of variable groups** is 1 if **Dependent variable grouping** is set to **Merge all variables** (the default), 2 if it is set to **Two groups**, and n_{TP} if it is set to **One variable per group**. Otherwise, the **Number of variable groups** setting can also be set to **User defined**.

MOTION SETTINGS

This section specifies the type of motion used in the model.

The **Motion type** can be either **No motion** or **Rotational motion** (the default). If **Rotational motion** is specified, it is possible to use feature that are specific for rotating machines, and it is possible to use the **Number of poles** n_{poles} input in order to synchronize different settings with each other. In particular, this input enables

synchronization of periodicity information between different nodes that are needed when just a sector of a full machine is modeled. For more information, see Typical Setup of Rotating Domain, Rotational Continuity Pair, and Rotational Periodicity.

DISCRETIZATION

Select the shape order for the **Magnetic vector potential** dependent variable — **Linear**, **Quadratic**, **Cubic**, **Linear serendipity** (the default), **Quadratic serendipity**, or **Cubic serendipity**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

🔍	Domain, Boundary, Edge, Point, and Pair Nodes for the Magnetic Machinery, Rotating, Time Periodic Interface

�	• *Permanent Magnet Motor with Campbell Diagram*: Application Library path **ACDC_Module/Devices,_Motors_and_Generators/ pm_motor_2d_campbell_diagram**

Domain, Boundary, Edge, Point, and Pair Nodes for the Magnetic Machinery, Rotating, Time Periodic Interface

The Magnetic Machinery, Rotating, Time Periodic Interface has these boundary, edge, pair, and point nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

💡	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

These domain nodes are described in this section:

- Exterior Boundaries
- Free Space
- Initial Values
- Coil
- Laminated Core

- Magnetic Potential (Pair)
- Perfect Magnetic Conductor (Pair)
- Rotating Domain
- Rotational Continuity Pair
- Rotational Periodicity

> **!** In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

The following nodes are described in the **Magnetic Fields** interface and, for **Multiphase Winding**, in the **Rotating Machinery, Magnetic** interface:

- External Current Density
- Force Calculation
- Magnet
- Magnetic Insulation

- Magnetic Potential
- Multiphase Winding
- Passive Conductor
- Perfect Magnetic Conductor

Initial Values

The **Initial Values** node adds an initial value for the out-of-plane component of the magnetic vector potential.

INITIAL VALUES

Enter a value or expression for the initial value of the **Magnetic vector potential out-of-plane** A_{0Z} (SI unit: Wb/m).

Free Space

The **Free Space** node is the default domain feature in the **Magnetic Machinery, Rotating, Time Periodic** interface. It adds the Ampère's law in free space and assumes that all domains have a uniform relative magnetic permeability of one.

Laminated Core

The **Laminated Core** node is used to model linear or nonlinear soft magnetic materials that contain no induced currents. The two available options for the **Magnetization model** setting are **B-H Curve** and **Relative permeability**, and are described for the Ampère's Law feature in the **Magnetic Fields** interface. It is also possible to specify the **Loss model**, which defines the **Volumetric loss density cycle average** (SI unit: W/m^3). It can be given as a **User defined** input, or, if **B-H Curve** is chosen as the **Magnetization model**, via **Steinmetz** or **Bertotti** coefficients. More information on loss models can be found in the documentation for the Loss Calculation feature.

Coil

The **Coil** node can be used to model coils, cables, and other devices subjected to an external lumped excitation current. The excitation is transformed into local quantities (electric fields and currents densities). The Coil feature supports only a **Single conductor** model, for which all the inputs are described in the Coil section in the **Magnetic Fields** interface. In addition, it is also possible to specify the **Loss Model** as either **From resistive heating** or **User defined**. More information on loss models can be found in the Loss Calculation section.

Exterior Boundaries

The **Exterior Boundaries** node is the default boundary condition in the **Magnetic Machinery, Rotating, Time Periodic** interface and adds a boundary condition in 2D models that constrains the out-of-plane magnetic vector potential to zero. **Exterior Boundaries** is therefore a special case of the **Magnetic Potential** boundary condition. One difference between the two is that **Exterior Boundaries** can only be applied on external boundaries.

Rotating Domain

The **Rotating Domain** node specifies which domains are rotating, relating the mechanical rotation α to the time periodic phase angle ϕ_{TP}. This node works in conjunction with the **Rotational Continuity Pair** node. The main input in this node is the **Rotation angle during time periodic cycle** α_{TP} (SI unit: rad), which may be user defined or based on the **Time periodic rotation** and **Rotation direction** settings.

If **Time periodic rotation** is set to **Synchronous (pole pair)**, the rotation angle is determined by the number of poles n_{poles} as $\alpha_{TP} = 4\pi/n_{poles}$. This is due to the fact that most of the fields in a synchronous machine are repeating after that angle.

If **Time periodic rotation** is set to **Full mechanical revolution**, the rotation angle becomes $\alpha_{TP} = 2\pi$ if **Rotation direction** is set to **Counterclockwise**, or $\alpha_{TP} = -2\pi$ if **Rotation direction** is set to **Clockwise**. This means that the specified regions rotate a complete turn in the specified direction during the time periodic frequency.

It is also possible to select **Nonrotating**, which results in $\alpha_{TP} = 0$. Finally, select **User defined** in order to manually enter a value of α_{TP}. The default expression is `if(mmtp.NoPoles==0,2*pi,4*pi/mmtp.NoPoles)` which indicates a full rotation counterclockwise if the number of poles defined at the physics level is zero, or a pole pair synchronous counterclockwise rotation if the number of poles is set to a finite value. It is also possible to override this value, for example when simulating asynchronous machines.

Once α_{TP} is defined, the mechanical rotation angle at a given frame is assumed to be proportional to the time periodic phase, as shown below.

$$\alpha = \alpha_{TP}\frac{\phi_{TP}}{2\pi}$$

Rotational Periodicity

The **Rotational Periodicity** node defines periodicity or antiperiodicity between two boundaries of a rotating magnetic machine where only a single sector is modeled. This feature works in conjunction with the **Rotating Domain** and **Rotational Continuity Pair** features. More details can be found in Typical Setup of Rotating Domain, Rotational Continuity Pair, and Rotational Periodicity.

Rotational Continuity Pair

Use the **Rotational Continuity Pair** node, selected from the **Pairs** menu, to specify a pair where the periodic condition should be applied to the magnetic vector potential between different parts that are rotating at different velocities. This feature works in conjunction with the **Rotating Domain** feature and, in the case of a sector, the **Rotational Periodicity** feature. More details can be found in Typical Setup of Rotating Domain, Rotational Continuity Pair, and Rotational Periodicity.

When the **Rotational Continuity Pair** is used, the phase dependent axial torque exerted by the two adjacent rotating parts is available in result evaluation on the form `<phystag>.<featuretag>.Tax_tpph` (for example, `mmtp.rcon1.Tax_tpph` if the tag of the physics interface is `mmtp` and the tag of the **Rotational Periodicity** feature is `rcon1`). More details on the structure of available variables can be found in Feature Specific Variables and Accessing the Solution at Different Phases. The torque calculation follows a procedure similar to that of Arkkio's method, which is described in the Arkkio's Method section.

Magnetic Potential (Pair)

The **Magnetic Potential** node from the **Pairs** menu adds independent values for the out-of-plane component of the magnetic vector potential on the source and the destination of a selected pair.

Perfect Magnetic Conductor (Pair)

Add the **Perfect Magnetic Conductor** node under **Pairs** in order to add a perfect magnetic conductor condition on both source and destination of a pair.

Typical Setup of Rotating Domain, Rotational Continuity Pair, and Rotational Periodicity

In Figure 5-6 and Figure 5-7 it is shown how typical conditions should be set for a rotating machine where either the full machine or a sector is drawn in the geometry. For the sake of simplicity, a synchronous machine with an external stator and an internal rotor with 10 poles is used in this example. Furthermore, the stator configuration is such that the minimal sector corresponds to one fifth of the whole geometry.

In this example, the features that should be added in both cases are as follows:

- The region inside of the cyan line should have a different **Rotating Domain** node, with **Time periodic rotation** set to either **Synchronous (pole pair)** or **User defined** so that $\alpha_{TP} = 4\pi/n_{poles} = 72°$.
- The cyan and red lines correspond to the source and destination selections for the pair where the **Rotational Continuity Pair** feature should be applied.
  - For such a pair, the source and destination should be either full circles or integer fractions of one. If an integer fraction is defined, a variable is introduced for the number of sectors that make up the full machine.
  - Both the source and the destination should be adjacent to only one single geometry component.
  - Multiple **Rotational Continuity Pair** nodes are allowed for modeling machines with multiple rotors.
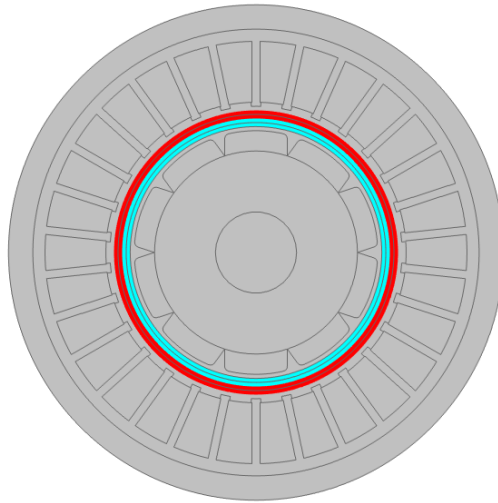
Figure 5-6: Regions where different features need to be applied for rotating motion, in the case where the full machine is modeled.

If a sector of the whole machine is drawn, a **Rotational Periodicity** node should also be added with a selection corresponding to the green line in Figure 5-7.

When modeling a single sector, it is important to specify if the symmetry is periodic or antiperiodic. If the **Rotational periodicity** type is set to **From physics**, this is done by

specifying the **Number of poles** in the **Motion Settings** section at the physics level. If the number of poles is an even multiple of the detected number of sectors, continuity is assumed. Conversely, if the number of poles is an odd multiple of the detected number of sectors, antiperiodicity is going to be assumed. The number of detected sectors is displayed along with the type of periodicity in the bottom of the **Motion Settings** section. The default value of the number of poles is zero, which results in periodicity being assumed.
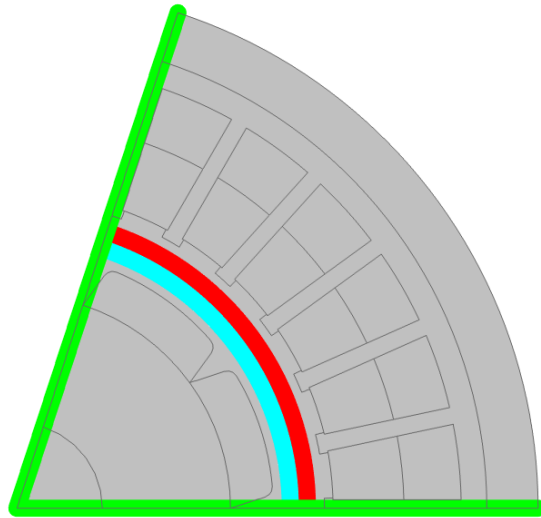
Figure 5-7: Regions where different features need to be applied for rotating motion, in the case where a single sector of the machine is modeled.

The button **Add Rotational Boundary Features** in the **Motion Settings** section automatically adds one **Rotational Continuity Pair** node for each pair that is geometrically consistent with two circular sliding objects. It also adds the corresponding **Rotational Continuity Pair** node if a single sector is drawn.

Feature Specific Variables and Accessing the Solution at Different Phases

When time periodic problems are solved, the solution contains variables that may have different values at different phase values.

In postprocessing, it is possible to use the built-in phase variable that can be specified as an input in a dataset. Variables in the physics interface that depend on the phase value typically end with the suffix _tpph. For example, the magnetic flux density norm, as a function of phase, can be accessed by writing mmtp.normB_tpph (assuming that the physics interface tag is mmtp).

It is also possible to access the value of a variable at a particular time by using the suffix _tpn, where n is an integer between 0 and the number of time frames n_{TP} - 1. For example, the magnetic flux density norm at phase zero is mmtp.normB_tp0 and the same variable at the phase $\pi/2$ is mmtp.normB_tp4 (assuming also that $n_{TP} = 16$).

Phase variables exist, for example, for the following quantities:

• Current density and electric field.
• Magnetic flux density, magnetic field, and out-of-plane magnetic vector potential.
• Feature specific postprocessing quantities such as voltages and force densities.
• Lumped and integral quantities such as magnetic coenergy or axial torque.

Since forces and torques are specific to the node they originate from, some extra details must be taken into account for their variable names:

• The total axial torque is added for each **Rotational Continuity Pair** node, so the variable contains the tag of that node. For example, mmtp.rcon1.Tax_tpph.
• The **Force Calculation** node distinguishes forces on different objects by an extra suffix that is specified as an input in the node. For example, the normal outward Maxwell stress tensor as a function of phase in the X direction becomes mmtp.nTX_1_tpph (given that the **Force name** input is 1).

When phase-dependent variables are defined, a number of related variables become available by substituting the _tpph suffix. In Table 5-7, available variables and operators are shown, using the X-component of the magnetic flux density as an example.

TABLE 5-7: POSSIBLE SUFFIXES AND OPERATORS FOR VARIABLES IN THE MAGNETIC MACHINERY, ROTATING, TIME PERIODIC INTERFACE.

SUFFIX	DESCRIPTION	EXAMPLE	OPERATOR
_tpph	Function of phase	mmtp.BX_tpph	mmtp.BX(phase)
_tp0	Time frame 1	mmtp.BX_tp0	mmtp.BX(0)
_tp1	Time frame 2	mmtp.BX_tp1	mmtp.BX(2*pi/mmtp.nTP)
_tpavg	Time periodic average	mmtp.BX_tpavg	mmtp.BX_fft(0)
_tprms	Time periodic root mean square	mmtp.BX_tprms	n.a

TABLE 5-7: POSSIBLE SUFFIXES AND OPERATORS FOR VARIABLES IN THE MAGNETIC MACHINERY, ROTATING, TIME PERIODIC INTERFACE.

SUFFIX	DESCRIPTION	EXAMPLE	OPERATOR
_tpdev	Time periodic standard deviation	mmtp.BX_tpdev	n.a
_fft1	Fourier component 1	mmtp.BX_fft1	mmtp.BX_fft(1)
_fft2	Fourier component 2	mmtp.BX_fft2	mmtp.BX_fft(2)

Axial torque and force density variables from the **Rotational Continuity Pair** and **Force Calculation** features are also equipped with versions giving the maximum and minimum values over a time period.

In Table 5-7, the last column shows the definition of operators that can be used for accessing the variables at a given phase, instead of the phase value specified in the dataset. These operators are available both during model setup and in postprocessing, where they can be found in the **Functions** section under **Definitions** in plot menus. It is important to note that the phase variable has a specific value (typically 0) in the physics. For accessing the phase at each time frame, the variable <phystag>.phaseTP can be used. However, this is not a true variable and cannot be plotted or included in other variable definitions. It may be used in the following inputs:

- **Relative permeability** for **Laminated Core** and **Passive Conductor**.
- **Current density out-of-plane** for **External Current Density**.
- **Magnetic vector potential out-of-plane** for **Initial Values** and **Magnetic Potential**.
- **Volumetric loss density** for **Passive Conductor**, **External Current Density**, **Magnet**, **Laminated Core**, and **Multiphase Winding**.

As an example, given that the physics interface tag is mmtp, it is possible to apply a current density with a DC bias of 1 A/mm^2 and a zero-phase harmonic ripple with an amplitude of 0.5 A/mm^2, by entering the expression 1[A/mm^2]+0.5[A/mm^2]*cos(mmtp.phaseTP) in the **External Current Density** feature.

6

The Electrical Circuit Interface

The Electrical Circuit interface, found under the **AC/DC** branch (⚡) when adding interfaces, has the equations for modeling electrical circuits with or without connections to a distributed field model, solving for the voltages, currents, and charges associated with the circuit elements.

In this chapter:

The Electrical Circuit Interface

The **Electrical Circuit (cir)** interface ( ), found under the **AC/DC** branch ( ) when adding a physics interface, is used to model currents and voltages in circuits including voltage and current sources, resistors, capacitors, inductors, and semiconductor devices. Models created with the Electrical Circuit interface can include connections to distributed field models. The physics interface supports stationary, frequency-domain and time-domain modeling and solves Kirchhoff's conservation laws for the voltages, currents, and charges associated with the circuit elements.

When this physics interface is added, it adds a default **Ground Node** feature and associates that with node zero in the electrical circuit.

![!]	Circuit nodes are nodes in the electrical circuit (electrical nodes) and should not be confused with nodes in the Model Builder tree of the COMSOL Multiphysics software. Circuit node names are not restricted to numerical values but can contain alphanumeric characters.

![icon]	*Inductor in an Amplifier Circuit*: Application Library path **ACDC_Module/Electromagnetics_and_Circuits/inductor_in_circuit**

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is cir.

RESISTANCE IN PARALLEL TO PN JUNCTIONS

For numerical stability, a large resistance is added automatically in parallel to the pn junctions in diodes and BJT devices. Enter a default value for the **Resistance in parallel to pn junctions** R_j (SI unit: Ω). The default value is $1 \cdot 10^{12}$ Ω.

CREATE UNIQUE NODES FOR NEW DEVICES

When this setting is selected (the default), newly added devices will be assigned unused node names. The devices will be disconnected from the rest of the circuit and the nodes should be updated to reflect the actual circuit connections. When this setting is deselected, new devices will be connected to the lowest-numbered nodes starting from 0.

🔍	• Theory for the Electrical Circuit Interface • Connecting to Electrical Circuits

ELECTRICAL CIRCUIT TOOLBAR

The following nodes are available from the **Electrical Circuit** ribbon toolbar (Windows users), **Electrical Circuit** context menu (Mac or Linux users), or right-click to access the context menu (all users):

💡	For step-by-step instructions and general documentation descriptions, this is the **Electrical Circuit** toolbar.

- Ground Node
- Voltmeter
- Ampère Meter
- Resistor
- Capacitor
- Inductor
- Voltage Source
- Current Source
- Diode
- Switch
- Voltage-Controlled Voltage Source[1]
- Voltage-Controlled Current Source[1]
- Current-Controlled Voltage Source[1]
- Current-Controlled Current Source[1]

- Subcircuit Definition
- Subcircuit Instance
- Mutual Inductance
- Transformer
- NPN BJT and PNP BJT[2]
- n-Channel MOSFET and p-Channel MOSFET[2]
- External I vs. U[3]
- External U vs. I[3]
- External I-Terminal[3]
- SPICE Circuit Import
- SPICE Circuit Export

[1] Selected from the **Dependent Sources** submenu when you right-click main node.

[2] Selected from the **Transistors** submenu when you right-click main node.

[3] Selected from the **External Couplings** submenu when you right-click main node.

Ground Node

The **Ground Node** (⏚) feature adds a ground node with the default node number zero to the electrical circuit. This is the default node in the Electrical Circuit interface. More ground nodes can be added but those must have unique node numbers and are by default given higher node numbers.

GROUND CONNECTION

Set the **Node name** for the ground node in the circuit. The convention is to use 0 (zero) for the ground node. If adding more ground nodes, each must have a unique node name (number).

Voltmeter

The **Voltmeter** (ⓥ) feature connects a voltmeter (voltage measurement device) between two nodes in the electrical circuit. A voltmeter behaves electrically as an open circuit. The voltmeter node adds a Probe sampling the voltage across it.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the resistor.

Ampère Meter

The **Ammeter** (ⓐ) feature connects an ammeter (current measurement device) between two nodes in the electrical circuit. An ammeter behaves electrically as a short circuit. The ammeter node adds a Probe sampling the current through it.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the resistor.

DEVICE PARAMETERS

Enter the **Resistance** of the resistor.

Resistor

The **Resistor** (⊏⊐) feature connects a resistor between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the resistor.

DEVICE PARAMETERS

Enter the **Resistance** of the resistor.

Capacitor

The **Capacitor** (⊣⊢) feature connects a capacitor between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the capacitor.

DEVICE PARAMETERS

Enter the **Capacitance** of the capacitor.

Inductor

The **Inductor** (⁰⁰⁰) feature connects an inductor between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the inductor.

DEVICE PARAMETERS

Enter the **Inductance** of the inductor.

Voltage Source

The **Voltage Source** (⊕) feature 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.

DEVICE PARAMETERS

Enter the **Source type** that should be adapted to the selected study type. It can be **General source**, **AC-source**, or a time-dependent **Sine source** or **Pulse source**. Depending on the choice of source, also specify the following parameters:

- For a General source, the **Voltage V_{src}** (default value: 1 V). General sources are active in Stationary, Time-Dependent and Frequency Domain studies.

- For an AC-source: the **Voltage V_{src}** (default value: 1 V) and the **Phase** Θ (default value: 0 rad). AC-sources are active in Frequency Domain studies only.

- For a sine source: the **Voltage V_{src}** (default value: 1 V), the **Offset V_{off}** (default value: 0 V), the **Frequency** (default value: 1 kHz), and the **Phase** Θ (default value: 0 rad). The sine sources are active in Time-Dependent studies and also in Stationary studies, providing that a value for t has been provided as a model parameter or global variable.

- For a pulse source: the **Voltage V_{src}** (default value: 1 V), the **Offset V_{off}** (default value: 0 V), the **Delay t_d** (default value: 0s), the **Rise time t_r** and **Fall time t_f** (default values: 0 s), the **Pulse width p_w** (default value: 1 μs), and the **Period T_{per}** (default value: 2 μs). The pulse sources are active in Time-Dependent studies and also in Stationary

studies, providing that a value for t has been provided as a model parameter or global variable.

All values are peak values rather than RMS.

	For the AC source, the frequency is a global input set by the solver. AC sources should be used in Frequency-domain studies only. Do not use the **Sine source** unless the model is time dependent.

Current Source

The **Current Source** (ϕ) feature 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 where the current flows through the source to the second node.

DEVICE PARAMETERS

Enter the **Source type** that should be adapted to the selected study type. It can be **General source**, **AC-source**, or a time-dependent **Sine source** or **Pulse source**. Depending on the choice of source, also specify the following parameters:

- For a General source, the **Current i_{src}** (default value: 1 A). General sources are active in Stationary, Time-Dependent and Frequency Domain studies.

- For an AC-source: the **Current i_{src}** (default value: 1 A) and the **Phase Θ** (default value: 0 rad). AC-sources are active in Frequency Domain studies only.

- For a sine source: the **Current i_{src}** (default value: 1 A), the **Offset i_{off}** (default value: 0 A), the **Frequency** (default value: 1 kHz), and the **Phase Θ** (default value: 0 rad). The sine sources are active in Time-Dependent studies and also in Stationary studies, providing that a value for t has been provided as a model parameter or global variable.

- For a pulse source: the **Current i_{src}** (default value: 1 A), the **Offset i_{off}** (default value: 0 A), the **Delay t_d** (default value: 0 s), the **Rise time t_r** and **Fall time t_f** (default values: 0 s), the **Pulse width p_w** (default value: 1 μs), and the **Period T_{per}** (default value: 2 μs). The pulse sources are active in Time-Dependent studies and also in Stationary

studies, providing that a value for t has been provided as a model parameter or global variable.

All values are peak values rather than RMS.

📝 | For the AC source, the frequency is a global input set by the solver. AC sources should be used in frequency-domain studies only. Do not use the **Sine source** unless the model is time dependent.

Voltage-Controlled Voltage Source

The **Voltage-Controlled Voltage Source** (◈) feature 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.

DEVICE PARAMETERS
There are two options to define the relationship between the control voltage and resulting voltage. The **Use gain** method defines the resulting voltage to be the control voltage multiplied by the gain. The **Custom expression** method can define the relationship with an arbitrary expression.

Voltage-Controlled Current Source

The **Voltage-Controlled Current Source** (◈) feature 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 where the current flows through the source to the second node.

There are two options to define the relationship between the control voltage and resulting current. The **Use gain** method defines the resulting current to be the control voltage multiplied by the gain (SI units: S). The **Custom expression** method can define the relationship with an arbitrary expression.

Current-Controlled Voltage Source

The **Current-Controlled Voltage Source** ( ) feature connects a current-controlled voltage source between two nodes in the electrical circuit. The input control current is the one flowing through 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.

DEVICE PARAMETERS

There are two options to define the relationship between the control current and resulting voltage. The **Use gain** method defines the resulting voltage to be the control current multiplied by the gain (SI units: Ω). The **Custom expression** method can define the relationship with an arbitrary expression.

Current-Controlled Current Source

The **Current-Controlled Current Source** ( ) feature 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 where the current flows through the source to the second node.

DEVICE PARAMETERS

There are two options to define the relationship between the control current and resulting current. The **Use gain** method defines the resulting current to be the control current multiplied by the gain. The **Custom expression** method can define the relationship with an arbitrary expression.

Switch

The **Switch** ( ) feature is used to connect or disconnect the conducting path in a circuit under specific conditions.

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 where the current flows through the source to the second node.

SWITCH CONDITIONS

There are three types of conditions, **Voltage controlled**, **Current controlled**, and **Custom expressions**. For each type of condition there are two conditions, one for turn on and one for turn off. The on condition is true if the **On condition** expression is larger than zero, while the off condition is true if the **Off condition** is less than zero.

Note: The state of the switch (on/off) is changed only when a condition goes from false to true and not when going from true to false. Thus, if the state is off and the on condition goes from false to true, the state will shift from off to on. On the other hand, if the state of the switch is on and the on condition goes from true to false, nothing will happen.

The **Initial state** list has three options, **Use on condition**, **Use off condition**, and **Boolean expression**. The two former options mean that the switch will have an initial state matching to the on or off condition. The third option makes the switch's initial state match a custom Boolean expression. Separating on, off, and initial states makes the switch more flexible and can support Schmitt-trigger style switches and various latches.

For the **Voltage controlled** switch, it is necessary to specify two nodes that defines the voltage `sens.v` that the switch state depends on. The conditions must be written as a function of this variable. Similarly, for the **Current controlled** switch it is necessary to specify a two-pin device that defines the current `sens.i` that the switch state depends on.

SWITCH PARAMETERS

When the switch is in the on state it has a nonzero resistance specified by the **On resistance** expression. For the off state no current flows through the switch (infinite resistance). There is also a transition time for the switch to turn on and off set by the

Switching time expression. The switch triggers an implicit event that updates a discrete state variable (with suffix `_state`).

Subcircuit Definition

The **Subcircuit Definition** (⊞) feature is used to define subcircuits, which can be inserted as devices into the main circuit using Subcircuit Instance nodes. Create the subcircuit by adding subnodes to the **Subcircuit Definition** node, either by using the **Physics** toolbar, or by right-clicking the **Subcircuit Definition**.

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 node. 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 node. The devices constituting the subcircuit should be connected only to the subcircuit's pins and to themselves.

INPUT PARAMETERS

To display this section, click the **Show More Options** button (⊙) and select **Advanced Physics Options**. Specify input parameters to a subcircuit that can be changed from a subcircuit instance. These input parameters can be used in all expression-style edit fields that affect the parameters of a device, for example, resistance, capacitance, and current gain. In this way, a subcircuit can represent a parameterized custom device model.

Subcircuit Instance

The **Subcircuit Instance** (⊡) feature represents an instance of a subcircuits defined by a Subcircuit Definition feature.

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.

INPUT PARAMETERS

To display this section, click the **Show More Options** button (⊙) and select **Advanced Physics Options**. Specify input parameters to a subcircuit that can be changed from a

subcircuit instance. These input parameters can be used in all expression-style edit fields that affect the parameters of a device, for example, resistance, capacitance, and current gain. In this way, a subcircuit can represent a parameterized custom device model.

NPN BJT and PNP BJT

The **NPN BJT** ( ) and the **PNP BJT** ( ) device models are large-signal models for bipolar junction transistors (BJT). It is an advanced device model and no thorough description and motivation of the many input parameters are attempted here. 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 **BJT** device. These represent the *collector*, *base*, and *emitter* nodes for the **NPN** transistor, and the *emitter*, *base*, and *collector* nodes for the **PNP** transistor.

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.

The interested reader is referred to Ref. 1 for more details on semiconductor modeling within circuits.

For an explanation of the **Model Parameters** see Bipolar Transistors.

---

n-Channel MOSFET and p-Channel MOSFET

The **n-Channel MOSFET** ( ) and the **p-Channel MOSFET** ( ) device models are large-signal models for, respectively, an n-Channel MOS field-effect transistor (MOSFET) and p-Channel MOSFET. These are advanced device models and no thorough description and motivation of the many input parameters are attempted here. Many device manufacturers provide model parameters for the MOSFET models. 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** or **p-Channel MOSFET** device. These represent the *drain*, *gate*, *source*, and *bulk* nodes, respectively.

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.

🔍	The interested reader is referred to Ref. 1 for more details on semiconductor modeling within circuits.
	For an explanation of the **Model Parameters** see MOSFET Transistors.

Mutual Inductance

The **Mutual Inductance** allows specifying a coupling between two existing Inductor features in the circuit. The mutual inductance of the coupling is

$$M = k \sqrt{L_1 L_2}$$

where k is the coupling factor and L_1 and L_2 are the inductances of the inductors.

DEVICE PARAMETERS

Enter values or expressions for the:

- **Coupling factor** k (dimensionless). The value must be between 0 and 1, and the default is 0.98.

- **First inductance** L_1 (SI unit: H) and **Second inductance** L_2 (SI unit: H). These must be set to two different Inductor features in the circuit.

Transformer

The **Transformer** feature represents either a combination of two Inductor and a Mutual Inductance features, or an ideal transformer.

NODE CONNECTIONS

Enter or edit the table in the **Node names** column for the **primary** and **secondary** node connections.

Choose a **Transformer model** — **Specify inductors** (the default) or **Ideal transformer**.

For **Specify inductors** enter values or expressions for the:

- **Coupling factor** k (dimensionless). The default is 0.98.

- **First inductance** L_1 (SI unit: H). The default is 1 mH.

- **Second inductance** L_2 (SI unit: H). The default is 1 mH.

For **Ideal transformer** enter values or expressions for the **Winding ratio** N_1/N_2 (dimensionless). The default is 10.

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 are attempted here. The interested reader is referred to Ref. 1 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.

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** (⟿) feature connects an arbitrary voltage measurement (for example, a circuit terminal or circuit port boundary or a coil domain from another physics interface) as a voltage 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.

EXTERNAL DEVICE

Enter the source of the **Voltage**. If circuit or current excited terminals or circuit ports are defined on boundaries or domains 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 because 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.

> **!** Except when coupling to a circuit terminal, circuit port, or coil, 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 tags are typically displayed within curly brackets {} in the Model Builder.

> **Q** Connecting to Electrical Circuits
>
> Nonlocal Couplings and Coupling Operators in the *COMSOL Multiphysics Reference Manual*

External U vs. I

The **External U vs. I** (⊡) feature connects an arbitrary current measurement (for example, a coil domain from another physics interface) as a current 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.

Specify the two **Node names** for the connecting nodes for the current source. The current flows from the first node to the second node.

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.

!	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 tags are typically displayed within curly brackets `{}` in the Model Builder.

Q	Connecting to Electrical Circuits
	Nonlocal Couplings and Coupling Operators in the *COMSOL Multiphysics Reference Manual*

External I-Terminal

The **External I-Terminal** (⌲) feature connects an arbitrary voltage-to-ground measurement (for example, a circuit terminal 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 node does not apply when coupling to inductive or electromagnetic wave propagation models because 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 node instead.

NODE CONNECTIONS

Set the **Node name** for the connecting node for the voltage assignment.

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.

> [!] • Except 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 tags are typically displayed within curly brackets {} in the Model Builder.
>
> • When connecting the finite element model between two circuit nodes that both are not grounded, the **External I-Terminal** node cannot be used. In this case, use External I vs. U or External U vs. I instead.

> Q Connecting to Electrical Circuits
>
> Nonlocal Couplings and Coupling Operators in the *COMSOL Multiphysics Reference Manual*.

SPICE Circuit Import

Right-click the **Electrical Circuit** ( ) feature 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.

> Q See SPICE Import about the supported SPICE commands.

SPICE Circuit Export

Right-click the **Electrical Circuit** ( ) feature node to export the current circuit to the SPICE netlist file format (select **Export 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 compatible circuit nodes are exported as SPICE devices

| | See SPICE Export for more details on the supported SPICE commands. |

Theory for the Electrical Circuit Interface

The Electrical Circuit Interface theory is discussed in this section:

• Electrical Circuit Modeling and the Semiconductor Device Models

• Bipolar Transistors

• MOSFET Transistors

• Diode

• Reference for the Electrical Circuit Interface

🔍	Connecting to Electrical Circuits

Electrical 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.

• The field model is used to get a better, more accurate description of a single device in the electrical circuit model.

• 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 possible to add nodes representing circuit elements directly to the Model Builder 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.

There are three more advanced large-signal semiconductor device features available in the Electrical Circuit interface. The equivalent circuits and the equations defining their nonideal circuit elements are described in this section. For a more complete treatise on semiconductor device modeling, see Ref. 1.

Bipolar Transistors

Figure 6-1 illustrates the equivalent circuit for the npn bipolar junction transistor.
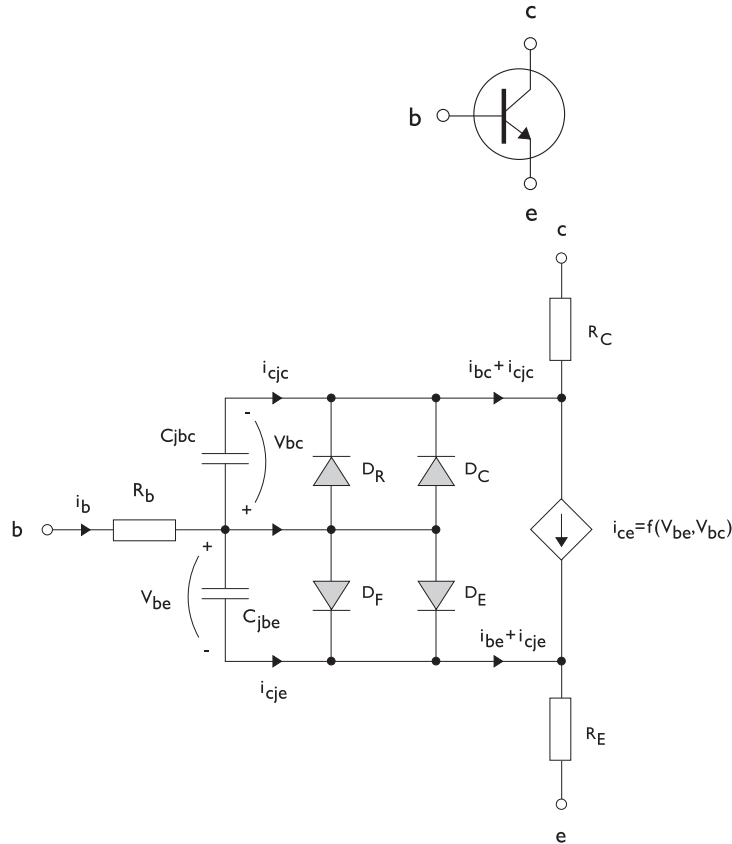
Figure 6-1: A circuit for the bipolar transistor.

The pnp transistor model is similar in all regards to the npn transistor, with the difference that the polarities of the currents and voltages involved are reversed. 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 6-1: 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 6-1: BIPOLAR TRANSISTOR MODEL PARAMETERS.

PARAMETER	DEFAULT	DESCRIPTION
I_S	10^{-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 Ω·m^2	Base resistance
R_{BM}	0 Ω·m^2	Minimum base resistance
R_C	0 Ω·m^2	Collector resistance
R_E	0 Ω·m^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

MOSFET Transistors

Figure 6-2 illustrates an equivalent circuit for the n-channel MOSFET transistor. The p-channel MOSFET transistor is treated similarly, but the polarities of the involved voltages are reversed.
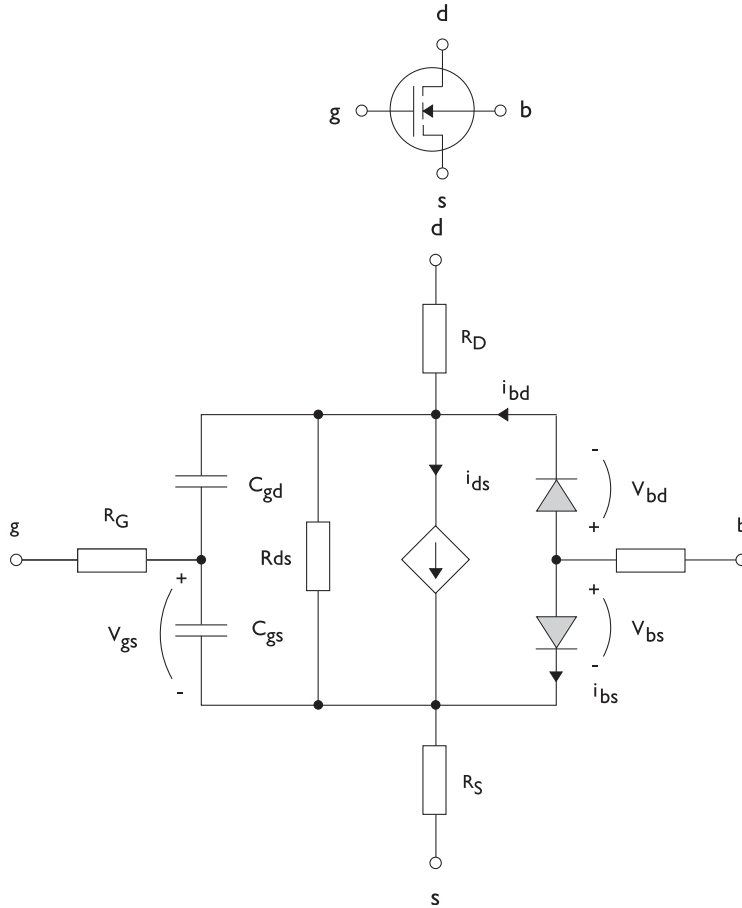
Figure 6-2: A circuit for the MOSFET transistor.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$i_{ds} = \begin{cases} \dfrac{W}{L}\dfrac{K_P}{2}(1 + \Lambda v_{ds})v_{ds}(2v_{th} - v_{ds}) & v_{ds} < v_{th} \\[2ex] \dfrac{W}{L}\dfrac{K_P}{2}(1 + \Lambda v_{ds})v_{th}^2 & v_{ds} \geq v_{th} \\[2ex] 0 & v_{ds} < v_{th} \leq 0 \end{cases}$$

$$v_{th} = v_{gs} - (V_{TO} + \Gamma(\sqrt{\Phi - v_{bs}} - \sqrt{\Phi}))$$

$$i_{bd} = I_S\left(e^{\frac{v_{bd}}{NV_T}} - 1\right)$$

$$i_{bs} = I_S\left(e^{\frac{v_{bs}}{NV_T}} - 1\right)$$

$$V_T = \frac{k_B T_{NOM}}{q}$$

There are also several capacitances between the terminals

$$C_{gd} = C_{gd0}W$$
$$C_{gs} = C_{gs0}W$$

$$C_{jbd} = C_{BD} \times \begin{cases} \left(1 - \dfrac{v_{bd}}{P_B}\right)^{-M_J} & v_{bx} < F_C P_B \\[2ex] (1 - F_C)^{-1 - M_J}\left(1 - F_C(1 + M_J) + M_J\dfrac{v_{bx}}{P_B}\right) & v_{bx} \geq F_C P_B \end{cases}$$

The model parameters are as follows:

TABLE 6-2: MOSFET 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

TABLE 6-2: MOSFET TRANSISTOR MODEL PARAMETERS.

PARAMETER	DEFAULT	DESCRIPTION
N	1	Bulk junction ideality factor
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)	1 $V^{0.5}$	Bulk threshold parameter
Φ (PHI)	0.5 V	Surface potential
Λ (LAMBDA)	0 1/V	Channel-length modulation

Diode

Figure 6-3 illustrates equivalent circuit for the diode.
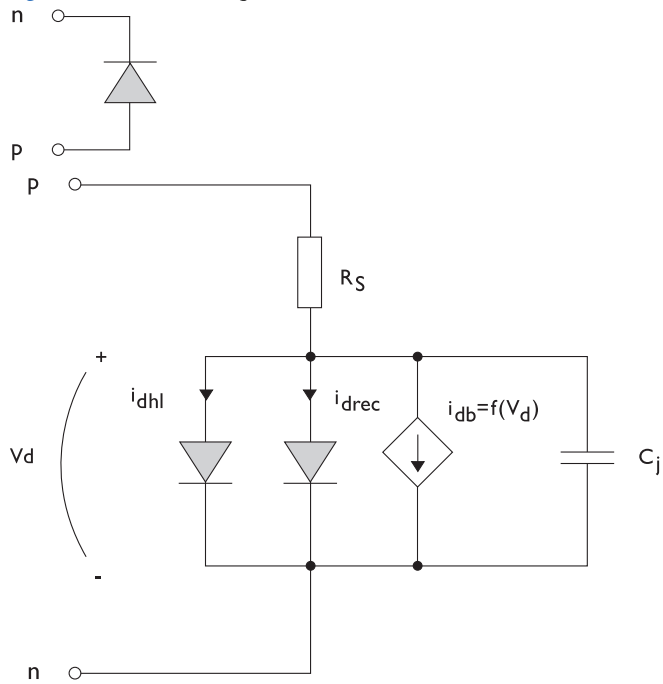
Figure 6-3: A circuit for the diode.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$i_d = i_{dhl} + i_{drec} + i_{db} + i_c$$

$$i_{dhl} = I_S\left(e^{\frac{v_d}{NV_T}} - 1\right)\frac{1}{\sqrt{1 + \frac{I_S}{I_{KF}}\left(e^{\frac{v_d}{NV_T}} - 1\right)}}$$

$$i_{drec} = I_{SR}\left(e^{\frac{v_d}{N_R V_T}} - 1\right)$$

$$i_{db} = I_{BV}e^{-\frac{v_d + B_V}{N_{BV}V_T}}$$

$$C_j = C_{J0} \times \begin{cases} \left(1 - \frac{v_d}{V_J}\right)^{-M} & v_d < F_C V_J \\[3mm] (1 - F_C)^{-1-M}\left(1 - F_C(1 + M) + M\frac{v_d}{V_J}\right) & v_d \ge F_C V_J \end{cases}$$

$$V_T = \frac{k_B T_{NOM}}{q}$$

where the following model parameters are required.

TABLE 6-3: 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

Reference for the Electrical Circuit Interface

1. P. Antognetti and G. Massobrio, *Semiconductor Device Modeling with Spice*, 2nd ed., McGraw Hill, 1993.

SPICE Import and Export

SPICE Import

The circuit definition in COMSOL Multiphysics adheres to the SPICE format developed at the University of California, Berkeley (see Ref. 1 for further information and references). SPICE netlists can be imported and the corresponding circuit nodes are generated in the COMSOL Multiphysics model. Most circuit simulators can export to this format or some version of it.

The Electrical Circuit interface supports the following device models:

TABLE 6-4: SUPPORTED SPICE DEVICE MODELS.

STATEMENT	DEVICE MODEL
R	Resistor
C	Capacitor
L	Inductor
V	Voltage Source
I	Current Source
E	Voltage-Controlled Voltage Source
F	Current-Controlled Current Source
G	Voltage-Controlled Current Source
H	Current-Controlled Voltage Source
D	Diode
Q	NPN BJT and PNP BJT
M	n-Channel MOSFET and p-Channel MOSFET
X	Subcircuit Instance

Statements corresponding to multiple devices are resolved by parsing the associated `.model` statement. The physics interface also supports the `.subckt` statement, which is represented in COMSOL by a Subcircuit Definition node, and the `.include` statement. SPICE commands are interpreted case-insensitively. The statement defining each device is also interpreted as the **Device name**.

According to SPICE specification, the first line in the netlist file is assumed to be the title of the netlist and it is ignored by the parser.

SPICE Export

The **SPICE Export** functionality creates a SPICE netlist file containing a description of the circuit represented by the physics interface. This functionality can be accessed from the physics interface context menu (right-click the physics interface node and select **Export SPICE Netlist**). After specifying a filename, the circuit is exported and messages from the export process display in the **Messages** window. During the export process, a series of operations are performed:

- In order to avoid conflicts, each component must be identified by a unique **Device name**. If one or more components have the same device name, the export operation fails and an error message is displayed. All characters in a **Device name** that are not letters, digits, or underscores are replaced by underscores.

- According to the SPICE specification, each circuit must have a node with name 0, which is assumed to be the only ground node. When exporting a circuit, any node with name 0 that is not connected to a **Ground** component is exported with a different node name. All nodes that are connected to a Ground components are exported as a merged node with name 0. The Messages window shows a log message if these operations are performed, showing the name of the renamed or merged nodes.

- All characters in node names that are not letters, digits, or underscores are replaced by underscores.

- Some components (most notably, the **External** components used to couple to other physics interfaces) cannot be exported to a SPICE netlist. These components are ignored during the export process, and a message is shown in the Messages window. Note that this can change the exported circuit, since some components are then missing.

- Subcircuit definitions are added as `.subckt` statements in the netlist. Semiconductor devices (such as MOSFETs, BJTs, and diodes) are exported as a SPICE device with a corresponding `.model` statement.

The title of the exported netlist file is the model's filename, and the time, date, and version of COMSOL Multiphysics is added as a comment in the netlist file.

Reference

1. https://en.wikipedia.org/wiki/SPICE

7

Multiphysics Interfaces and Couplings

The AC/DC Module contains predefined multiphysics interfaces to facilitate easy setup of models with the most commonly occurring couplings. When adding a multiphysics interface, two or more physics interfaces are added to the model, together with one or more multiphysics couplings.

Given that your model already contains two interfaces that support a certain multiphysics coupling, you can also add this coupling separately. Some multiphysics couplings can only be added this way, and are not part of a multiphysics interface.

In this chapter:

- Overview of Electromagnetic Heating Interfaces
- Overview of Electromagnetics-Structure Interaction Interfaces
- Overview of Electromagnetics–Fluid Interaction Interfaces
- Overview of Electromagnetics-Particle Interaction Interfaces
- Overview of Multiphysics Couplings

Overview of Electromagnetic Heating Interfaces

The **Model Wizard** and the **Add Physics** window contain a number of multiphysics interfaces for modeling electromagnetic heating, all available under the **Electromagnetic Heating** branch ( ). The branch includes multiphysics interfaces that combine electromagnetics with heat transfer or heat transfer and structural mechanics. The multiphysics interfaces for electromagnetic heating are described in the subsequent sections:

- The Joule Heating Interface ( ): It combines all features from the Electric Currents interface with the Heat Transfer interface to model resistive heating and heating due to dielectric losses. The predefined multiphysics couplings add the electromagnetic power dissipation as a heat source, and the electromagnetic material properties can depend on the temperature.

- The Joule Heating and Thermal Expansion Interface ( ): It combines thermal, electric, and structural multiphysics effects. The predefined interaction adds the electromagnetic losses from the electric current as a heat source. In addition, the temperature from the Heat Transfer in Solids interface acts as a thermal load for the Solid Mechanics interface, causing thermal expansion. It requires either the Structural Mechanics Module or the MEMS Module license.

- The Induction Heating Interface ( ): It combines all features from the Magnetic Fields interface in the time-harmonic formulation with the Heat Transfer interface to model induction and eddy current heating. The predefined multiphysics couplings add the electromagnetic power dissipation as a heat source, and the electromagnetic material properties can depend on the temperature. This physics interface is based on the assumption that the magnetic cycle time is short compared to the thermal time scale (adiabatic assumption).

- The Pyroelectricity Interface ( ): It combines Heat Transfer in Solids and Electrostatics interfaces together with the constitutive relationships required to model pyroelectricity. Both the direct pyroelectric and inverse electrocaloric effects can be modeled. It requires the MEMS Module license.

The Joule Heating Interface

The **Joule Heating** interface ( ) is used to model resistive heating and, depending on additional licensed products, dielectric heating in devices where inductive effects are negligible; that is, when the skin depth is much larger than the studied device. This multiphysics interface adds an Electric Currents interface and a Heat Transfer in Solids interface. The multiphysics couplings add the electromagnetic power dissipation as a heat source, and the electromagnetic material properties can depend on the temperature.

Depending on the licensed products, stationary modeling, time-domain modeling are supported in all space dimensions. In addition, combinations of frequency-domain modeling for the Electric Currents interface and stationary modeling for the Heat Transfer in Solids interface, called frequency-stationary and frequency-transient modeling, are supported.

When a predefined **Joule Heating** interface is added from the **AC/DC** > **Electromagnetic Heating** or **Heat Transfer** > **Electromagnetic Heating** branch ( ) of the **Model Wizard** or **Add Physics** windows, **Electric Currents** and **Heat Transfer in Solids** interfaces are added to the Model Builder. In addition, The Multiphysics Branch is added, which automatically includes the multiphysics coupling feature **Electromagnetic Heating**.

On the Constituent Physics Interfaces
The Electric Currents interface computes electric field, current, and potential distributions in conducting media under conditions where inductive effects are negligible; that is, when the skin depth is much larger than the studied device. Depending on the licensed products, time and frequency domain formulations that account for capacitive effects are also provided. The Electric Currents interface solves a current conservation equation based on Ohm's law using the scalar electric potential as the dependent variable.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation. A Heat Transfer in Solids model is active by default on all domains. All functionality for including other domain types, such as a fluid domain, is also available. The temperature equation defined in solid domains

corresponds to the differential form of the Fourier's law that may contain additional contributions like heat sources.

> In previous versions of COMSOL Multiphysics, a specific physics interface called Joule Heating was added to the Model Builder. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see Multiphysics Modeling Workflow.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURE

When physics interfaces are added using the predefined couplings, for example **Joule Heating**, specific settings are included with the physics interfaces and the coupling features.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single **Electric Currents** and **Heat Transfer in Solids** interfaces are added, the COMSOL Multiphysics software adds an empty **Multiphysics** node. When you right-click this node, you can choose **Electromagnetic Heating** from the available coupling features, but the modified settings are not included.

TABLE 7-1: MODIFIED SETTINGS FOR A JOULE HEATING INTERFACE.

PHYSICS INTERFACE OR COUPLING FEATURE	MODIFIED SETTINGS (IF ANY)
Electric Currents	No changes.
Heat Transfer in Solids	No changes.
Electromagnetic Heating	The **Domain Selection** is the same as that of the participating physics interfaces.
	The **Boundary Selection** is the same as the exterior and interior boundaries of the **Domain Selection** of the participating physics interfaces.
	The corresponding **Electric Currents** and **Heat Transfer in Solids** interfaces are preselected in the Coupled Interfaces section.

PHYSICS INTERFACE AND COUPLING FEATURE

Coupling Feature
The Electromagnetic Heating multiphysics coupling node is described in this section.

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (macOS or Linux users), or right-click to access the context menu (all users).

💡 | In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using.

• The available physics features for the heat transfer interfaces are listed in the sections Domain Features, Boundary Features, and Edge and Point Features.

• For information about the available physics features for the Electric Currents interface, see The Electric Currents Interface.

💡 | If you have an add-on module, such as the Heat Transfer Module or AC/DC Module, there are additional specialized physics nodes available and described in the individual module documentation.

📖 | *Thermal Microactuator Simplified*: Application Library path **COMSOL_Multiphysics/Multiphysics/thermal_actuator_simplified**

Electromagnetic Heating

The **Electromagnetic Heating** multiphysics coupling (🔥) adds the source term Q_e (SI unit: W/m^3) to account for resistive heating in the heat equation:

$$\rho C_p \frac{\partial T}{\partial t} - \nabla \cdot (k \nabla T) = Q_e \qquad (7\text{-}1)$$

The resistive heating (ohmic heating) due to the electric current is

$$Q_e = \mathbf{J} \cdot \mathbf{E}$$

where **J** is the current density (SI unit: A/m^2), and **E** is the electric field strength (SI unit: V/m).

In addition, it maps the electromagnetic surface losses as a heat source on the boundary (SI unit: W/m^2) in the heat transfer part of the model.

The **Label** is the default multiphysics coupling name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling in the model) is emh1.

DOMAIN SELECTION

When nodes are added from the context menu, you can select **Manual** (the default) from the **Selection** list to choose specific domains to define the electromagnetic heat source or select **All domains** as needed.

When **Electromagnetic Heating** is added as an effect of adding a **Joule Heating** or **Joule Heating and Thermal Expansion** interface, the selection is the same as for the participating interfaces.

Only domains that are active in the physics interfaces selected in the **Coupled Interfaces** section can be selected.

BOUNDARY SELECTION

When nodes are added from the context menu, you can select **Manual** (the default) from the **Selection** list to choose specific boundaries to define the electromagnetic boundary heat source or select **All boundaries** as needed.

When **Electromagnetic Heating** is added as an effect of adding a **Joule Heating** or **Joule Heating and Thermal Expansion** interface, the selection is the same as the exterior and interior boundaries of the **Domain Selection** of the participating physics interfaces.

Only boundaries that are active in the physics interfaces selected in the **Coupled Interfaces** section can be selected.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. By default, the applicable physics interface is selected in the **Electromagnetic** list to apply the **Heat transfer** to its physics interface to establish the coupling.

The default values depend on how the coupling node is created.

• If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (macOS and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.

• If it is added automatically when a multiphysics interface is selected in the **Model Wizard** or **Add Physics** window, then the two participating physics interfaces are selected.

You can also select **None** from either list to uncouple the **Electromagnetic Heating** node from a physics interface. If the physics interface is removed from the **Model Builder** (for example, **Heat Transfer in Solids** is deleted), then the **Heat transfer** list defaults to **None** as there is nothing to couple to.

Click the **Go to Source** button (📑) to move to the main physics interface node for the selected physics interface.

> 📝 If a physics interface is deleted and then added to the model again, then in order to reestablish the coupling, you need to choose the physics interface again from the **Heat transfer** or **Electromagnetic** lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present physics interface. See Multiphysics Modeling Workflow.

The Joule Heating and Thermal Expansion Interface

The **Joule Heating and Thermal Expansion** interface (🔲) combines thermal, electric, and structural multiphysics effects. The predefined interaction adds the electromagnetic losses from the electric current as a heat source. In addition, the temperature from the Heat Transfer in Solids interface acts as a thermal load for the Solid Mechanics interface, causing thermal expansion. Optionally, effects from thermoelastic damping and inelastic dissipation can be used as heat sources in the heat transfer analysis.

You can use this multiphysics coupling for coupled thermal, electrical, and structural analysis of, for example, the movement of some actuator, where an electric current causes a temperature increase, which in turn leads to a displacement through thermal expansion.

When a predefined **Joule Heating and Thermal Expansion** interface is added from the **Structural Mechanics** branch (⬚) of the **Model Wizard** or **Add Physics** windows, Solid Mechanics, Electric Currents, and Heat Transfer in Solids interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, containing the multiphysics coupling nodes Thermal Expansion and Electromagnetic Heating.

On the Constituent Physics Interfaces
- The Electric Currents interface is described in The Electric Currents Interface.
- The Solid Mechanics interface is described in The Solid Mechanics Interface.
- The Heat Transfer in Solids interface is described in The Heat Transfer in Solids Interface.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined couplings, for example **Joule Heating and Thermal Expansion**, specific settings are included with the physics interfaces and the coupling features.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single Solid Mechanics, Electric Currents, and Heat Transfer in Solids interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics** node. You can then choose from the available coupling features, **Thermal Expansion**, and **Electromagnetic Heating**, but the modified settings are not included.

TABLE 7-2: MODIFIED SETTINGS FOR A JOULE HEATING AND THERMAL EXPANSION INTERFACE.

PHYSICS INTERFACE OR COUPLING FEATURE	MODIFIED SETTINGS
Solid Mechanics	For the Solid Mechanics interface, under Structural Transient Behavior, the **Structural transient behavior** is set to **Quasi static**.
Electric Currents	No changes.
Heat Transfer in Solids	No changes.
Thermal Expansion	The **Domain Selection** is the same as that of the participating physics interfaces. The corresponding Solid Mechanics and Heat Transfer in Solids interfaces are preselected in the Thermal Expansion section.
Electromagnetic Heating	The **Domain Selection** is the same as that of the participating physics interfaces. The corresponding Electric Currents and Heat Transfer in Solids interfaces are preselected in the Electromagnetic Heating section.

If you have an add-on module, such as the Heat Transfer Module or AC/DC Module, there are additional specialized physics nodes available and described in the individual module documentation.

The Induction Heating Interface

The **Induction Heating** interface ( ) is used to model induction heating and eddy current heating. This multiphysics interface adds a Magnetic Fields interface and a Heat Transfer in Solids interface. The multiphysics couplings add the electromagnetic power dissipation as a heat source, and the electromagnetic material properties can depend on the temperature.

Depending on the licensed products, stationary modeling and time-domain modeling are supported in 2D and 3D. In addition, combinations of frequency-domain modeling for the Magnetic Fields interface and stationary modeling for the Heat Transfer in Solids interface, called frequency-stationary and, similarly, frequency-transient modeling, are supported.

When a predefined **Induction Heating** interface is added from the **AC/DC** > **Electromagnetic Heating** or **Heat Transfer** > **Electromagnetic Heating** branch ( ) of the **Model Wizard** or **Add Physics** windows, **Magnetic Fields** and **Heat Transfer in Solids** interfaces are added to the Model Builder. A **Multiphysics Couplings** node is also added, which automatically includes the multiphysics coupling feature **Electromagnetic Heating**.

On the Constituent Physics Interfaces
The Magnetic Fields interface is used to compute magnetic field and induced current distributions in and around coils, conductors, and magnets. Depending on the licensed products, stationary, frequency-domain, and time-domain formulations are supported in 2D and 3D. Note that the frequency and time domain formulations become ill-posed when approaching the static limit. One may extend the useful frequency range downward by adding a low electric conductivity. The Magnetic Fields interface solves Maxwell's equations formulated using the magnetic vector potential and, optionally for s, the scalar electric potential as the dependent variables.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation. A Heat Transfer in Solids model is active by default on all domains. All functionality for including other domain types, such as a fluid domain, is also available. The temperature equation defined in solid domains

corresponds to the differential form of Fourier's law that may contain additional contributions like heat sources.

📝 In previous versions of COMSOL Multiphysics, a specific physics interface called Induction Heating was added to the Model Builder. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURE

When physics interfaces are added using the predefined couplings, for example **Induction Heating**, specific settings are included with the physics interfaces and the coupling features.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single **Magnetic Fields** and **Heat Transfer in Solids** interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose **Electromagnetic Heating** from the available coupling features but the modified settings are not included.

💡 Coupling features are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

TABLE 7-3: MODIFIED SETTINGS FOR AN INDUCTION HEATING INTERFACE.

PHYSICS INTERFACE OR COUPLING FEATURE	MODIFIED SETTINGS (IF ANY)
Magnetic Fields	No changes.
Heat Transfer in Solids	No changes.
Electromagnetic Heating	The **Domain Selection** is the same as that of the participating physics interfaces.
	The **Boundary Selection** is the same as the exterior and interior boundaries of the **Domain Selection** of the participating physics interfaces.
	The corresponding **Magnetic Fields** and **Heat Transfer in Solids** interfaces are preselected in the Coupled Interfaces section.

	A side effect of adding physics interfaces one at a time is that four study types — Frequency-Stationary; Frequency-Transient; Frequency-Stationary, One-Way Electromagnetic Heating; and Frequency-Transient, One-Way Electromagnetic Heating — are not available for selection until *after* at least one coupling feature is added. In this case, it is better to initially not add any study at all, then add the coupling features to the **Multiphysics** node, and lastly, open the **Add Study** window and add a study sequence below the **Preset Studies for Selected Multiphysics** heading.

PHYSICS INTERFACES AND COUPLING FEATURE

	Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

The Electromagnetic Heating coupling feature node is described in this section.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Magnetic Fields Interface are listed in the section Domain, Boundary, Point, and Pair Nodes for the Magnetic Fields Interface.
- See The Heat Transfer Interfaces in the *COMSOL Multiphysics Reference Manual* for information about the available physics features for heat transfer.

	Inductive Heating of a Copper Cylinder: Application Library path **ACDC_Module/Electromagnetic_Heating/inductive_heating**

Electromagnetic Heating

The **Electromagnetic Heating** multiphysics coupling node 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

$$Q_e = Q_{rh} + Q_{ml}$$

where the resistive losses are

$$Q_{rh} = \frac{1}{2}\text{Re}(\mathbf{J} \cdot \mathbf{E}^*)$$

and the magnetic losses are

$$Q_{ml} = \frac{1}{2}\text{Re}(i\omega\mathbf{B} \cdot \mathbf{H}^*)$$

In addition, it maps the electromagnetic surface losses as a heat source on the boundary (SI unit: W/m^2) in the heat transfer part of the model.

SETTINGS

The **Label** is the default multiphysics coupling name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling in the model) is emh1.

DOMAIN SELECTION

When nodes are added from the context menu, you can select **Manual** (the default) from the **Selection** list to choose specific domains to define the electromagnetic heat source or select **All domains** as needed.

When **Electromagnetic Heating** is added as an effect of adding an **Induction Heating** interface, the selection is the same as for the participating interfaces.

Only domains that are active in the physics interfaces selected in the **Coupled Interfaces** section can be selected.

BOUNDARY SELECTION

When nodes are added from the context menu, you can select **Manual** (the default) from the **Selection** list to choose specific boundaries to define the electromagnetic boundary heat source or select **All boundaries** as needed.

When **Electromagnetic Heating** is added as an effect of adding an **Induction Heating** interface, the selection is the same as the exterior and interior boundaries of the **Domain Selection** of the participating physics interfaces.

Only boundaries that are active in the physics interfaces selected in the **Coupled Interfaces** section can be selected.

COUPLED INTERFACES

This section defines the physics involved in the **Electromagnetic Heating** multiphysics coupling. By default, the applicable physics interface is selected in the **Electromagnetic** list to apply the **Heat transfer** to its physics interface to establish the coupling.

You can also select **None** from either list to uncouple the **Electromagnetic Heating** node from a physics interface. If the physics interface is removed from the **Model Builder**, for example **Heat Transfer in Solids** is deleted, then the **Heat transfer** list defaults to **None** as there is nothing to couple to.

	If a physics interface is deleted and then added to the model again, and in order to reestablish the coupling, you need to choose the physics interface again from the **Heat transfer** or **Electromagnetic** lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present physics interface. See Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

Overview of Electromagnetics-Structure Interaction Interfaces

The **Model Wizard** and the **Add Physics** window contain a number of multiphysics interfaces for modeling of structure interaction with magnetic and electromagnetic fields, all available under the **Electromagnetics and Mechanics** branch ( ) within the **AC/DC** branch ( ). Most of the multiphysics interfaces require an AC/DC Module license and the addition of either a Structural Mechanics Module or a MEMS Module license. This section gives an overview introduction to these multiphysics interfaces. For detailed descriptions, click the hyperlinks of the interface names listed below:

- The Electromechanics, Solid Interface ( ): It combines Solid Mechanics and Electrostatics with a moving mesh functionality to model the deformation of electrostatically actuated structures.

- The Electromechanics, Boundary Elements Interface ( ): It combines the functionality of Solid Mechanics and Electrostatics to model the deformation of electrostatically actuated structures. The coupling is a boundary load caused by the Maxwell Stress at the interface of the solid domains and nonsolid voids, where the electric field is computed using the boundary element method. The backward coupling to Electrostatics is due to the deformations of the boundaries.

- The Piezoelectricity, Solid Interface ( ): It combines Solid Mechanics and Electrostatics together with the constitutive relationships required to model piezoelectrics. Both the direct and inverse piezoelectric effects can be modeled and the piezoelectric coupling can be formulated using the strain-charge or stress-charge forms.

- The Piezoelectricity, Layered Shell Interface ( ): It combines structural layered shells and electric currents in shells together with the constitutive relationships required to model the direct and inverse piezoelectric effects. The piezoelectric coupling can be formulated using either the strain-charge or stress-charge forms.

- The Pyroelectricity Interface ( ): It combines Solid Mechanics, Electrostatics, and Heat Transfer in Solids together with the constitutive relationships required to model piezoelectric applications, in which the temperature variation and electric charge are coupled. This interface requires the MEMS Module license.

- The Electrostriction Interface ( ): It combines Solid Mechanics and Electrostatics together with the constitutive relationships required to model electrostriction in electrostatically actuated structures in regimes when the electric polarization can be assumed to vary linear with the applied electric field. Both the direct and inverse electrostrictive effects can be modeled.

- The Ferroelectroelasticity Interface ( ): It combines Solid Mechanics and Electrostatics together with the constitutive relationships required to model nonlinear electromechanical interaction in ferroelectric and piezoelectric materials. Electric polarization in such materials depends nonlinearly on the applied electric field including possible hysteresis and saturation effects. In addition, the polarization and mechanical deformations in such materials can be strongly coupled due to the electrostriction effect.

- The Nonlinear Magnetostriction Interface ( ): It combines Solid Mechanics and Magnetic Fields together with the constitutive relationships required to model magnetostrictive materials and devices. Both the direct and inverse magnetostriction effects can be modeled.

- The Piezomagnetism Interface ( ): It combines Solid Mechanics and Magnetic Fields together with the constitutive relationships required to model linear magnetostrictive materials and devices. Both the direct and inverse piezomagnetic effects can be modeled.

- The Magnetomechanics Interface ( ): It combines Solid Mechanics and Magnetic Fields interfaces together with a moving mesh functionality. The interface can be used for modeling deformation of magnetically actuated structures, which includes interaction of magnetic fields with magnetic materials and current carrying elements such as coils and wires.

- The Magnetomechanics, No Currents Interface ( ): It combines Solid Mechanics and Magnetic Fields, No Currents interfaces with a moving mesh functionality to model the deformation of magnetostatically actuated structures. The interface is suitable in cases when induction and conduction effects such as eddy currents are negligible within the structure.

- The Magnetic–Elastic Interaction in Rotating Machinery Interface ( ): It combines Solid Mechanics and Rotating Machinery, Magnetic interfaces with a moving mesh functionality. The interface can be used for modeling deformation of rotating magnetic machinery structures such as electric motors and generators. Using this interface, you can compute how stress originated from air gap forces are distributed in both stator and rotor of an electric motor. Some of the applications are magnetic bearings, unbalanced rotors.

- The Magnetic–Rigid-Body Interaction in Rotating Machinery Interface (⊙): It combines Multibody Dynamics and Rotating Machinery, Magnetic interfaces with a moving mesh functionality. The interface can be used for modeling deformation and large rotational motions of rotating magnetic machinery structures such as electric motors and generators. Using this interface, you can compute how stress originated from air gap forces are distributed in both stator and rotor of an electric motor. Some of the applications are magnetic bearings, unbalanced rotors.

- The Piezoresistivity, Domain Currents Interface (▦): It combines the Solid Mechanics and Electric Currents interfaces to simulate the piezoresistive effect. The physics interface is appropriate for situations when the thicknesses of the conducting and piezoresistive layers are resolved by the mesh.

- The Piezoresistivity, Boundary Currents Interface (▦): It combines the Solid Mechanics interface with electric current flow within a thin layer of the solid to model piezoresistance. The interface is appropriate for situations in which the thickness of the piezoresistive layer is much less than that of the structural layer.

- The Piezoresistivity, Shell Interface (▦): It combines structural shells with electric current flow on a thin layer within the shell to simulate the piezoresistive effect. The physics interface is used when the structural layer is thin enough to be treated by the Shell interface, but the conducting and piezoresistive layers are still much thinner than the structural layers. This physics interface requires the addition of the Structural Mechanics Module.

Overview of Electromagnetics–Fluid Interaction Interfaces

The **Model Wizard** and the **Add Physics** window contain a number of multiphysics interfaces for modeling fluid interaction with magnetic and electromagnetic fields, all available under the **Electromagnetics and Fluids** branch (🧲) within the **AC/DC** branch (🔻). This section gives an overview introduction to these multiphysics interfaces. The multiphysics interfaces for electromagnetics-fluid interaction are described in the subsequent sections:

- The Magnetohydrodynamics Interface (🧲): Combines the **Magnetic and Electric Fields** and **Laminar Flow** to model the interaction of magnetic fields and conducting fluids in 3D. The magnetic fields are solved with the **Magnetic and Electric Fields** interface, which includes the solving of a current conservation equation.

- The **Magnetohydrodynamics, In-Plane Currents** interface (🧲): Available in 2D and 2D axisymmetric, it has the same settings as The Magnetohydrodynamics Interface.

- The **Magnetohydrodynamics, Out-of-Plane Currents** interface (🧲): Available in 2D and 2D axisymmetric, it shares most of the settings with The Magnetohydrodynamics Interface. The difference is that the magnetic field is computed with the **Magnetic Fields** interface, which does not solve a current conservation equation. This interface is used here since it is computationally cheaper than the **Magnetic and Electric Fields** interface.

The Magnetohydrodynamics Interface

The **Magnetohydrodynamics** interface ( ) is used to model the interaction between moving conducting fluids and magnetic fields. This multiphysics interface adds the **Magnetic and Electric Fields** and the **Laminar Flow** interfaces. The interface can be used to model a variety of magnetohydrodynamics phenomena in astrophysics, geophysics, metallurgy, and electrolytes. To model equilibrium discharges, use the dedicated **Equilibrium Discharges** interfaces in the Plasma Module.

Stationary and transient modeling are supported in 2D, 2D axisymmetric, and 3D. Also supported are combinations of frequency-domain modeling for the Magnetic Fields interface and stationary/transient modeling for the Laminar Flow interface, called frequency-stationary and frequency-transient modeling.

When a predefined **Magnetohydrodynamics** interface is added from the **AC/DC** > **Electromagnetics and Fluids** branch ( ) of the **Model Wizard** or **Add Physics** windows, **Magnetic and Electric Fields** and **Laminar Flow** interfaces are added to the **Model Builder**. In addition, The Multiphysics Branch is added, which automatically includes the multiphysics coupling feature **Magnetohydrodynamics**.

On the Constituent Physics Interfaces
The Magnetic and Electric Fields interface computes electric field, current, and magnetic field distributions in conducting fluids. By default, it solves Ampère's Law and current conservation equations in the computation domain.

The Laminar Flow interface solves for conservation of energy, mass, and momentum in fluids.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURE

When physics interfaces are added using the predefined couplings, for example **Magnetohydrodynamics**, specific settings are included with the physics interfaces and the coupling features.

In contrast, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if separate **Electromagnetics and Fluids** and **Laminar Flow** interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics** node. When you right-click

this node, you can choose **Magnetohydrodynamics** from a list of available coupling features but the modified settings are not included.

TABLE 7-4: MODIFIED SETTINGS FOR A JOULE HEATING INTERFACE.

PHYSICS INTERFACE OR COUPLING FEATURE	MODIFIED SETTINGS (IF ANY)
Magnetic and Electric Fields	The discretization of the dependent variables is set to linear in 3D.
Laminar Flow	No changes.
Magnetohydrodynamics	The **Domain Selection** is the same as that of the participating physics interfaces
	The corresponding **Magnetic and Electric Fields** and **Laminar Flow** interfaces are preselected in the Coupled Interfaces section.

PHYSICS INTERFACE AND COUPLING FEATURE

Coupling Feature
The Magnetohydrodynamics multiphysics coupling node is described in the next section.

Physics Interface Features
Physics nodes are available from the **Physics** ribbon toolbar (on Windows), the **Physics** context menu (on macOS and Linux), or by right-clicking to access the context menu (all platforms).

Magnetohydrodynamics

The **Magnetohydrodynamics** ( ) multiphysics coupling feature is used to model the interaction of magnetic fields and conducting fluids. It couples one of the magnetic field interfaces and the **Laminar Flow** interface. The supported magnetic field interfaces are the **Magnetic Fields**, **Magnetic and Electric Fields**, **Magnetic Field Formulation**, and **Rotating Machinery, Magnetic** interfaces in the AC/DC Module. By default, the coupling feature passes the Lorentz force $\mathbf{F} = \mathbf{J} \times \mathbf{B}$ from the magnetic field interface to the **Laminar Flow** interface and the electromotive force (the induced electric field) $\mathbf{E} = \mathbf{v} \times \mathbf{B}$ from the **Laminar Flow** interface to the magnetic field interface.

The **Magnetohydrodynamics** feature is available in 2D, 2D axisymmetric, and 3D geometries and is allowed for both Stationary and Time Dependent studies (including Frequency-Stationary and Frequency-Transient studies). When the deformation of the conducting fluids cannot be neglected (that is, when topology changes are important),

the **Moving Mesh** feature (added from the **Definitions**) is usually used together with the **Magnetohydrodynamics** feature.

!	• The **Magnetohydrodynamics** feature cannot be used to model magnetizable fluids (ferrohydrodynamics). In magnetizable fluids, the magnetization forces can be much larger than the Lorentz force. In such cases, a user-defined multiphysics coupling is required.
	• The difference between the **Magnetic and Electric Fields** interface and the **Magnetic Fields** interface is that the former solves the current conservation equation explicitly, while the latter assumes the conservation of current implicitly. In 3D magnetohydrodynamics modeling, the most general approach is to use the **Magnetic and Electric Fields** interface to solve the magnetic field. The **Magnetic Fields**, **Magnetic Field Formulation**, and **Rotating Machinery, Magnetic** interfaces can be useful in specific cases.

SETTINGS

The **Label** is the multiphysics coupling feature name. The default **Label** (for the first multiphysics coupling feature in the model) is Magnetohydrodynamics 1.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is mhd1.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Electromagnetic** and **Fluid flow** lists include all applicable physics interfaces.

When the Magnetohydrodynamics node is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (macOS and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.

You can also select **None** from either list to uncouple the Magnetohydrodynamics node from a physics interface. If the physics interface is removed from the **Model Builder**, for

example **Solid Mechanics** is deleted, then the list defaults to **None** as there is nothing to couple to.

> If a physics interface is deleted and then added to the model again, then in order to reestablish the coupling, you need to choose the physics interface again from the lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present interface. See Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

COUPLED SETTINGS

The coupling behavior can be easily controlled by selecting or clearing the **Include Lorentz force** and **Include electromotive force** checkboxes. For example, if **Include Lorentz force** is selected and **Include electromotive force** is cleared, only the Lorentz force is considered and the electromotive force is neglected. This is usually valid when the magnetic Reynolds number is very low.

There are several library examples that demonstrate its usage:

> *Hartmann Boundary Layer*: Application Library path **ACDC_Module/ Electromagnetics_and_Fluids/hartmann_boundary_layer**
>
> *Magnetohydrodynamics Pump*: Application Library path **ACDC_Module/ Electromagnetics_and_Fluids/magnetohydrodynamics_pump**

Overview of Electromagnetics-Particle Interaction Interfaces

The **Model Wizard** and the **Add Physics** window contain a number of multiphysics interfaces for modeling electromagnetics-particle interaction, all available under the **Particle Tracing** branch (⚛) within the **AC/DC** branch (⚡). The multiphysics interfaces under this branch requires a Particle Tracing Module license. This section gives an overview introduction to these multiphysics interfaces. For detailed descriptions, click the hyperlinks of the interface names listed below:

- The Charged Particle Tracing Interface (⚛): Use it to model charged particle orbits under the influence of electromagnetic forces. In addition, it can also model two-way coupling between the particles and fields. Some typical applications are particle accelerators, vacuum tubes and ion implanters. The physics interface supports time-domain modeling only in 2D and 3D. The interface solves the equation of motion for charged particles subjected to electromagnetic forces.

- The Particle–Field Interaction, Nonrelativistic Interface (⚛): Combines the Charged Particle Tracing interface with the Electrostatics interface. The Electric Particle–Field Interaction multiphysics coupling feature is added automatically. The Particle–Field Interaction, Nonrelativistic interface is used to model beams of charged particles at nonrelativistic speeds. The particles generate a space charge density term as they propagate through domains. The space charge density is then used as a source term in the Electrostatics interface, and the resulting electric force on the particles is computed.

- The Particle–Field Interaction, Relativistic Interface (⚛): Combines the Charged Particle Tracing, Electrostatics, and Magnetic Fields physics interfaces. The Electric Particle–Field Interaction and Magnetic Particle–Field Interaction multiphysics coupling features are added automatically. The Particle–Field Interaction, Relativistic interface is used to model beams of relativistic charged particles. The particles generate space charge density and current density terms as they propagate through domains. The space charge density and current density are then used to compute electric and magnetic forces, respectively, which are exerted on the particles.

Overview of Multiphysics Couplings

The AC/DC Module contains a number of Multiphysics Coupling features that can couple electromagnetic domains to heat-transfer media, structures, and charged particles to model a variety of multiphysics problems. Several of the coupling features are used in predefined multiphysics interfaces but can also be added manually. Except for the coupling features between AC/DC interfaces, most of the coupling features require additional licenses of other modules such as the Structural Mechanics Module.

This section summarizes all Multiphysics Coupling features related to the AC/DC Module. For detailed descriptions, click the hyperlinks of the feature names listed below:

- Coupling features between AC/DC Interfaces:

    - Electric Scalar-Scalar Potential Coupling (▭): Assures continuity of the electric potential across boundaries between The Electrostatics Interface and The Electrostatics, Boundary Elements Interface.

    - Magnetic Scalar-Scalar Potential Coupling (▭): Assures continuity of the magnetic scalar potential across boundaries between The Magnetic Fields, Currents Only Interface and The Magnetic Fields, No Currents, Boundary Elements Interface.

    - Magnetic Vector-Scalar Potential Coupling (▭): Assures continuity of the magnetic flux density across boundaries between The Magnetic Fields Interface and The Magnetic Fields, No Currents, Boundary Elements Interface.

    - Electric Currents Solid to Shell Via Boundary (▱): Assures continuity of the electric potential across boundaries between The Electric Currents Interface and The Electric Currents in Shells Interface or The Electric Currents in Layered Shells Interface.

    - Magnetic Fields Formulation Magnetic Fields No Currents Coupling (▭): Assures continuity of the normal magnetic flux density and the tangential

magnetic field across boundaries between The Magnetic Field Formulation Interface and The Magnetic Fields, No Currents Interface.

- Coupling features in predefined Electromagnetic Heating interfaces:
  - Electromagnetic Heating ( ) in The Joule Heating Interface.
  - Electromagnetic Heating ( ) and Thermal Expansion ( ) in The Joule Heating and Thermal Expansion Interface.
  - Electromagnetic Heating ( ) in The Induction Heating Interface.
  - Pyroelectricity ( ) in The Pyroelectricity Interface and in The Piezoelectricity and Pyroelectricity Interface.
- Coupling features in predefined Electromagnetics–Structure Interaction Interfaces:
  - Electromechanics ( ) in The Electromechanics, Solid Interface.
  - Electromechanics, Boundary Elements ( ) in The Electromechanics, Boundary Elements Interface.
  - Piezoelectricity ( ) in The Piezoelectricity, Solid Interface.
  - Piezoelectricity, Layered ( ) in The Piezoelectricity, Layered Shell Interface.
  - Electrostriction ( ) in The Electrostriction Interface and in The Ferroelectroelasticity Interface.
  - Piezomagnetism ( ) in The Piezomagnetism Interface.
  - Nonlinear Magnetostriction ( ) in The Nonlinear Magnetostriction Interface.
  - Magnetomechanics ( ) in The Magnetomechanics Interface.
  - Magnetomechanics ( ) in The Magnetomechanics, No Currents Interface.
  - Magnetic Forces, Rotating Machinery ( ) in The Magnetic–Elastic Interaction in Rotating Machinery Interface and The Magnetic–Rigid-Body Interaction in Rotating Machinery Interface.
  - Piezoresistivity, Domain Currents ( ) in The Piezoresistivity, Domain Currents Interface.
  - Piezoresistivity, Boundary Currents ( ) in The Piezoresistivity, Boundary Currents Interface.
  - Piezoresistivity, Shell ( ) in The Piezoresistivity, Shell Interface.
- Coupling features in predefined Electromagnetics–Fluid Interaction Interfaces:
  - Magnetohydrodynamics ( ) in The Magnetohydrodynamics Interface.

- Coupling features in predefined Electromagnetics–Particle Interaction Interfaces:
  - Electric Particle Field Interaction ( ) in The Particle–Field Interaction, Nonrelativistic Interface.
  - Electric Particle Field Interaction ( ) and Magnetic Particle–Field Interaction ( ) in The Particle–Field Interaction, Relativistic Interface.
- Other Multiphysics Coupling features:
  - Electromagnetic Heating, Layered Shell ( ): Use it to account for electromagnetic surface losses as a heat source in the heat equation in layered materials represented by boundaries.
  - Thermoelectric Effect, Layered Shell ( ): Use it to account for a Peltier heat source or sink in layered materials represented by boundaries where electrical and thermal models are defined.

8

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, see the glossary in the *COMSOL Multiphysics Reference Manual*. 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 **D** and **E** fields and between the **B** and **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 See *electric currents point dipole*, *electrostatics point dipole*, and *magnetic fields electric dipole*.

electric currents point dipole Two equal and opposite point current sources $+q$ and $-q$ separated a short distance d. The electric current dipole moment is given by $\mathbf{p} = q\mathbf{d}$, where $\mathbf{d}$ is a vector going from $-q$ to $+q$.

electrostatics point 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.

magnetic fields electric dipole The limiting case when the length d of a current filament carrying uniform current I approaches zero while maintaining the product between I and d.

Nedelec's edge element See *vector element*.

phasor A complex function of space representing a sinusoidally varying quantity.

quasistatic 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