AC/DC Module User’s Guide

© 1998–2011 COMSOL

Protected by U.S. Patents 7,519,518; 7,596,474; and 7,623,991. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/sla) and may be used or copied only under the terms of the license agreement.

COMSOL, COMSOL Desktop, COMSOL Multiphysics, and LiveLink are registered trademarks or trademarks of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

Version: May 2011

Part No. CM020101
Contents

Chapter 1: Introduction

About the AC/DC Module ............................................. 14
What Can the AC/DC Module Do? ................................. 14
AC/DC Module Physics Interface Guide ......................... 14
AC/DC Module Study Availability .................................. 17
Where Do I Access the Documentation and Model Library? ........................................................................ 18
Typographical Conventions ........................................... 20
Overview of the User’s Guide ......................................... 22

Chapter 2: Review of Electromagnetics

Fundamentals of Electromagnetics ................................. 26
Maxwell’s Equations ..................................................... 26
Constitutive Relations ................................................... 27
Potentials .................................................................... 30
Reduced Potential PDE Formulations ............................ 30
Electromagnetic Energy ................................................ 31
The Quasi-Static Approximation and the Lorentz Term ........................................................................ 33
Material Properties ....................................................... 34
About the Boundary and Interface Conditions. ................. 36
Phasors .................................................................... 37

Electromagnetic Forces .................................................. 39
Overview of Forces in Continuum Mechanics .................... 39
Forces on an Elastic Solid Surrounded by Vacuum or Air ........................................................................ 41
Torque ...................................................................... 42
Forces in Stationary Fields ............................................. 43
Forces in a Moving Body ................................................. 46
Electromagnetic Energy and Virtual Work ....................... 49
Chapter 3: Modeling with the AC/DC Module

Preparing for Modeling 58
What Problems Can You Solve? 59
Selecting the Space Dimension for the Model Geometry 59
Simplifying the Geometry Using Boundary Conditions 62
Applying Electromagnetic Sources 63
Selecting a Study Type 63
Field Variables in 2D 64
Meshing and Solving 65

Infinite Elements 67
Modeling Unbounded Domains 67
Known Issues When Modeling Using Infinite Elements 70

Force and Torque Computations 72
Calculating Electromagnetic Forces and Torques 72
Model Examples—Electromagnetic Forces 73

Lumped Parameters 74
Calculating Lumped Parameters with Ohm’s Law 74
Calculating Lumped Parameters Using the Energy Method 76
Studying Lumped Parameters 77

Lumped Ports with Voltage Input 79
About Lumped Ports 79
Lumped Port Parameters 79

Special Calculations 51
Mapped Infinite Elements 51
Lumped Parameter Conversion 52

Electromagnetic Quantities 53

References for the AC/DC Interfaces 55
Chapter 4: The Electric Field Interfaces

The Electrostatics Interface
- Charge Conservation ........................................ 100
- Space Charge Density ........................................ 102
- Force Calculation. ........................................... 102
- Infinite Elements ............................................. 103
- Manual Scaling .............................................. 104
- Initial Values. ............................................... 105
- Boundary Conditions for the Electrostatics Interface .... 106
- Pairs for the Electrostatics Interface. ...................... 107
- Ground ...................................................... 107
- Electric Potential ........................................... 109
- Surface Charge Density ..................................... 110
- Dielectric Shielding ........................................ 111
- Terminal .................................................... 112
- Floating Potential .......................................... 113
- Displacement Field ......................................... 114
- Distributed Capacitance .................................... 115
- Periodic Condition ........................................... 116
- Zero Charge .................................................. 117
- Thin Low Permittivity Gap .................................. 117
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuity</td>
<td>118</td>
</tr>
<tr>
<td>Line Charge</td>
<td>119</td>
</tr>
<tr>
<td>Point Charge</td>
<td>120</td>
</tr>
<tr>
<td>Electrostatic Point Dipole</td>
<td>120</td>
</tr>
<tr>
<td>The Electric Currents Interface</td>
<td>122</td>
</tr>
<tr>
<td>Current Conservation</td>
<td>124</td>
</tr>
<tr>
<td>Archie's Law</td>
<td>127</td>
</tr>
<tr>
<td>External Current Density.</td>
<td>128</td>
</tr>
<tr>
<td>Current Source</td>
<td>129</td>
</tr>
<tr>
<td>Initial Values</td>
<td>130</td>
</tr>
<tr>
<td>Boundary Conditions for the Electric Currents Interface</td>
<td>131</td>
</tr>
<tr>
<td>Pairs for the Electric Currents Interface</td>
<td>132</td>
</tr>
<tr>
<td>Boundary Current Source</td>
<td>132</td>
</tr>
<tr>
<td>Normal Current Density</td>
<td>133</td>
</tr>
<tr>
<td>Distributed Impedance</td>
<td>135</td>
</tr>
<tr>
<td>Electric Shielding</td>
<td>136</td>
</tr>
<tr>
<td>Electric Insulation</td>
<td>139</td>
</tr>
<tr>
<td>Periodic Condition</td>
<td>140</td>
</tr>
<tr>
<td>Contact Impedance and Pair Contact Impedance</td>
<td>140</td>
</tr>
<tr>
<td>Sector Symmetry</td>
<td>142</td>
</tr>
<tr>
<td>Continuity</td>
<td>143</td>
</tr>
<tr>
<td>Line Current Source</td>
<td>143</td>
</tr>
<tr>
<td>Point Current Source</td>
<td>144</td>
</tr>
<tr>
<td>Electric Point Dipole</td>
<td>145</td>
</tr>
<tr>
<td>The Electric Currents, Shell Interface</td>
<td>146</td>
</tr>
<tr>
<td>Initial Values</td>
<td>148</td>
</tr>
<tr>
<td>Boundary Conditions for the Electric Currents, Shell Interface</td>
<td>148</td>
</tr>
<tr>
<td>Edge (3D) or Point (2D) Conditions</td>
<td>149</td>
</tr>
<tr>
<td>Theory of Electric Fields</td>
<td>151</td>
</tr>
<tr>
<td>Charge Relaxation Theory</td>
<td>151</td>
</tr>
<tr>
<td>Theory for the Electrostatics Interface</td>
<td>155</td>
</tr>
<tr>
<td>Electrostatics Equations</td>
<td>155</td>
</tr>
</tbody>
</table>
## Chapter 5: The Magnetic Field Interfaces

### The Magnetic Fields Interface

- **Ampère's Law** ........................................ 166
- **Gauge Fixing for A-field** .......................... 169
- **External Current Density** ......................... 170
- **Velocity (Lorentz Term)** .......................... 171
- **Multi-Turn Coil Domain** ........................... 172
- **Single-Turn Coil Domain** .......................... 174
- **Coil Group Domain** .................................. 176
- **Reversed Current Direction** ....................... 180
- **Initial Values** ....................................... 180
- **Boundary Conditions for the Magnetic Fields Interface** ........................................ 181
- **Pairs for the Magnetic Fields Interface** ........ 182
- **Magnetic Insulation** ................................ 182
- **Magnetic Field** ....................................... 184
- **Surface Current** ..................................... 184
- **Lumped Port** ......................................... 185
- **Magnetic Potential** .................................. 187
- **Impedance Boundary Condition** ................. 188
- **Perfect Magnetic Conductor** ...................... 190
- **Transition Boundary Condition** ................... 191
- **Thin Low Permeability Gap** ....................... 193
- **Periodic Condition** ................................ 194
Chapter 6: The Magnetic and Electric Fields Interface

The Magnetic and Electric Fields Interface

Ampère's Law and Current Conservation
Initial Values
Domain Conditions for the Magnetic and Electric Fields Interface
Boundary Conditions for the Magnetic and Electric Fields Interface
Point and Edge Conditions for the Magnetic and Electric Fields Interface
Edge Current

Theory for the Magnetic and Electric Fields Interface

Magnetostatics Equations
Frequency Domain Equations

Chapter 7: The Electrical Circuit Interface

The Electrical Circuit Interface

Ground Node
Resistor
Capacitor
Inductor
Voltage Source
Current Source
Voltage-Controlled Voltage Source
Voltage-Controlled Current Source
Current-Controlled Voltage Source
Current-Controlled Current Source
Subcircuit Definition
Subcircuit Instance
NPN BJT
n-Channel MOSFET
Chapter 8: Heat Transfer Branch

The Induction Heating Interface

Shared Feature Nodes for the Induction Heating Interface

Induction Heating Model

Electromagnetic Heat Source

Initial Values

Chapter 9: Materials

Material Library and Databases

About the Material Databases

About Using Materials in COMSOL

Opening the Material Browser

Using Material Properties
Chapter 10: Glossary

Glossary of Terms
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 AC/DC 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
- Where Do I Access the Documentation and Model Library?
- Typographical Conventions

What Can the AC/DC Module Do?

The AC/DC Module (the Module) is an optional package that extends the COMSOL Multiphysics® modeling environment. This Module contains a set of interfaces adapted to a broad category of electromagnetic simulations and it solves problems in the general areas of electrostatic fields, magnetostatic fields, and quasi-static fields.

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

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

COMSOL Multiphysics also has an interface to the MATLAB technical computing environment. If you have a MATLAB license, you can save it as a Model M-file—a script file that runs in MATLAB.

AC/DC Module Physics Interface Guide

The physics interfaces in the AC/DC Module form a complete set of simulation tools for electromagnetic field simulations. To select the right physics interface for describing the real-life physics you need to consider the geometric properties and the
time variations of the fields. 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 interface has a Tag which is of special importance when performing multiphysics simulations. This tag helps distinguish between physics interfaces and the variables defined by the interface have an underscore plus the physics interface tag appended to their names.

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

In 2D, in-plane and out-of-plane variants are available for problems with a planar symmetry as well as axisymmetric interfaces for problems with a cylindrical symmetry. When using an axisymmetric interface it is important to note that the horizontal axis represents the $r$ direction and the vertical axis the $z$ direction, and that you must create the geometry in the right half-plane (that is, for positive $r$ only). 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.

### Physics Interface Availability by Space Dimension

<table>
<thead>
<tr>
<th>Physics Interface</th>
<th>Icon</th>
<th>Tag</th>
<th>1D</th>
<th>1D Axi</th>
<th>2D</th>
<th>2D Axi</th>
<th>3D</th>
</tr>
</thead>
<tbody>
<tr>
<td>AC/DC</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Electrostatics*</td>
<td></td>
<td>es</td>
<td>✅</td>
<td>✅</td>
<td>✅</td>
<td>✅</td>
<td>✅</td>
</tr>
<tr>
<td>Electric Currents*</td>
<td></td>
<td>ec</td>
<td>✅</td>
<td>✅</td>
<td>✅</td>
<td>✅</td>
<td>✅</td>
</tr>
<tr>
<td>Electric Currents - Shell</td>
<td></td>
<td>ecs</td>
<td></td>
<td></td>
<td></td>
<td>✅</td>
<td>✅</td>
</tr>
<tr>
<td>Magnetic Fields*</td>
<td></td>
<td>mf</td>
<td>✅</td>
<td>✅</td>
<td>✅</td>
<td>✅</td>
<td>✅</td>
</tr>
<tr>
<td>Magnetic and Electric Fields</td>
<td></td>
<td>mef</td>
<td></td>
<td></td>
<td></td>
<td>✅</td>
<td>✅</td>
</tr>
</tbody>
</table>
### PHYSICS INTERFACE

<table>
<thead>
<tr>
<th>PHYSICS INTERFACE</th>
<th>ICON</th>
<th>TAG</th>
<th>1D</th>
<th>2D</th>
<th>3D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Magnetic Fields, No Currents</td>
<td><img src="image" alt="Magnetic Fields" /></td>
<td>mfn</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Rotating Machinery, Magnetic</td>
<td><img src="image" alt="Rotating Machinery" /></td>
<td>rmm</td>
<td></td>
<td>✓</td>
<td></td>
</tr>
<tr>
<td>Electrical Circuit</td>
<td><img src="image" alt="Electrical Circuit" /></td>
<td>cir</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Electromagnetic Heating</td>
<td><img src="image" alt="Electromagnetic Heating" /></td>
<td>ih</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
</tbody>
</table>

* An enhanced interface is one that is included with the base COMSOL package but has added functionality for this Module.

### SEE ALSO
- What Can the AC/DC Module Do?
- AC/DC Module Study Availability
- Where Do I Access the Documentation and Model Library?
- Typographical Conventions
# AC/DC Module Study Availability

<table>
<thead>
<tr>
<th>PHYSICS INTERFACE</th>
<th>TAG</th>
<th>DEPENDENT VARIABLES</th>
<th>FIELD COMPONENTS</th>
<th>PRESET STUDIES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>MAGNETIC FIELD</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>ELECTRIC FIELD</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>MAGNETIC POTENTIAL</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>CURRENT DENSITY</td>
<td>STATIONARY</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>TIME DEPENDENT</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>FREQUENCY DOMAIN</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>SMALL-SIGNAL ANALYSIS, FREQUENCY DOMAIN</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>FREQUENCY STATIONARY</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>FREQUENCY TRANSIENT</td>
<td></td>
</tr>
<tr>
<td>Electrostatics</td>
<td>es</td>
<td>V</td>
<td>x</td>
<td>√</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>y</td>
<td>√</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>z</td>
<td></td>
</tr>
<tr>
<td>Electric Currents</td>
<td>ec</td>
<td>V</td>
<td>x</td>
<td>√</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>y</td>
<td>√</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>z</td>
<td>√</td>
</tr>
<tr>
<td>Electric Currents, Shell</td>
<td>ecs</td>
<td>V</td>
<td>x</td>
<td>√</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>y</td>
<td>√</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>z</td>
<td></td>
</tr>
<tr>
<td>Magnetic Fields</td>
<td>mf</td>
<td>A</td>
<td>x</td>
<td>√</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>y</td>
<td>√</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>z</td>
<td>√</td>
</tr>
<tr>
<td>Magnetic and Electric Fields</td>
<td>mef</td>
<td>V, A</td>
<td>x</td>
<td>√</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>y</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>z</td>
<td></td>
</tr>
<tr>
<td>Magnetic Fields, No Currents</td>
<td>mfnc</td>
<td>V_m</td>
<td>x</td>
<td>√</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>y</td>
<td></td>
</tr>
<tr>
<td>Rotating Machinery, Magnetic</td>
<td>rmm</td>
<td>A</td>
<td>x</td>
<td>√</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>y</td>
<td></td>
</tr>
<tr>
<td>Electrical Circuit</td>
<td>cir</td>
<td>not applicable</td>
<td></td>
<td>√</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
CHAPTER 1: INTRODUCTION

SEE ALSO

- What Can the AC/DC Module Do?
- AC/DC Module Physics Interface Guide
- Where Do I Access the Documentation and Model Library?
- Typographical Conventions
- Solver Studies and Study Types in the COMSOL Multiphysics User’s Guide
- Study Types in the COMSOL Multiphysics Reference Guide

Where Do I Access the Documentation and Model Library?

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

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

<table>
<thead>
<tr>
<th>PHYSICS INTERFACE</th>
<th>TAG</th>
<th>DEPENDENT VARIABLES</th>
<th>FIELD COMPONENTS</th>
<th>PRESET STUDIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Induction Heating</td>
<td>ih</td>
<td>A, T, J</td>
<td>x, y, z</td>
<td></td>
</tr>
</tbody>
</table>

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

**Custom studies are also available based on the interface, for example, Eigenfrequency and Eigenvalue.
**Note:** If you are working directly from a PDF on your computer, the blue links do not work to open a model or documentation referenced in a different user guide. However, if you are using the online help desk in COMSOL Multiphysics, these links work to other modules, model examples, and documentation sets.

**The Documentation**

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

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

- Click the buttons on the toolbar or
- Select Help>Documentation ( ) or Help>Dynamic Help ( ) from the main menu and then either enter a search term or look under a specific module in the documentation tree.

**The Model Library**

Each model comes with a theoretical background and step-by-step instructions to create the model. The models are available in COMSOL as MPH-files that you can open for further investigation. Use both the step-by-step instructions and the actual models as a template for your own modeling and applications. SI units are used to describe the relevant properties, parameters, and dimensions in most examples, but other unit systems are available.

To open the Model Library, select View>Model Library ( ) from the main menu, and then search by model name or browse under a Module folder name. If you also want to review the documentation explaining how to build a model, select the model and click Model PDF or the Dynamic Help button ( ) to reach the PDF or HTML version, respectively. Alternatively, select Help>Documentation in COMSOL and search by name or browse by Module.

If you have feedback or suggestions for additional models for the library (including those developed by you), feel free to contact us at info@comsol.com.
COMSOL WEB SITES
Main corporate web site: http://www.comsol.com/
Worldwide contact information: http://www.comsol.com/contact/
Online technical support main page: http://www.comsol.com/support/
COMSOL Support Knowledge Base, your first stop for troubleshooting assistance, where you can search for answers to any COMSOL questions: http://www.comsol.com/support/knowledgebase/

CONTACTING COMSOL BY EMAIL
For general product information, contact COMSOL at info@comsol.com.
To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and case number is sent to you by email.

COMSOL COMMUNITY
On the COMSOL web site, you find a user community at http://www.comsol.com/community/. The user community includes a discussion forum, a model exchange, news postings, and a searchable database of papers and presentations.

Typographical Conventions
All COMSOL user guides use a set of consistent typographical conventions that should make it easy for you to follow the discussion, realize what you can expect to see on the screen, and know which data you must enter into various data-entry fields.

In particular, you should be aware of these conventions:
• Click text highlighted in blue to go to other information in the PDF. When you are using the online help desk in COMSOL Multiphysics, these links also work to other Modules, model examples, and documentation sets.
• A boldface font of the shown size and style indicates that the given word(s) appear exactly that way on the COMSOL Desktop (or, for toolbar buttons, in the corresponding tooltip). For example, the Model Builder window ( ) is often referred to and this is the window that contains the model tree. As another example, the instructions might say to click the Zoom Extents button ( ), and this boldface
font means that you can expect to see a button with that label (when you hover over the button with your mouse) on the COMSOL Desktop.

- The names of other items on the COMSOL Desktop that do not have direct labels contain a leading uppercase letter. For instance, we often refer to the Main toolbar—the horizontal bar containing several icons that are displayed on top of the user interface. However, nowhere on the COMSOL Desktop, nor the toolbar itself, includes the word “Main”.

- The forward arrow symbol > indicates selecting a series of menu items in a specific order. For example, **Options > Results** is equivalent to: From the **Options** menu, select **Results**.

- A **Code** (monospace) font indicates you are to make a keyboard entry in the user interface. You might see an instruction such as “Enter (or type) 1.25 in the **Current density** edit field.” The monospace font also is an indication of programming code. or a variable name. An italic **Code** (monospace) font indicates user inputs and parts of names that can vary or be defined by the user.

- An italic font indicates the introduction of important terminology. Expect to find an explanation in the same paragraph or in the Glossary. The names of other user guides in the COMSOL documentation set also have an italic font.

**The Difference Between Nodes, Buttons, and Icons**

- **Node**: A node is located in the **Model Builder** and has an icon image to the left of it. Right-click a node to open a Context Menu and to perform actions.

- **Button**: Click a button to perform an action. Usually located on a toolbar (the Main toolbar or the **Graphics** window toolbar, for example), or in the upper right corner of a **Settings** window.

- **Icon**: An icon is an image that displays on a window (for example, the **Model Wizard** or **Model Library**) or displays in a Context Menu when a node is right-clicked. Sometimes selecting an icon from a node’s Context Menu adds a node with the same image and name, sometimes it simply performs the action indicated (for example, **Delete**, **Enable**, or **Disable**).
Overview of the User’s Guide

The AC/DC Module User’s Guide gets you started with modeling using COMSOL Multiphysics. The information in this guide is specific to this Module. Instructions how to use COMSOL in general are included with the COMSOL Multiphysics User’s Guide. As detailed in the section Where Do I Access the Documentation and Model Library? this information is also searchable from the COMSOL Multiphysics software Help menu.

TABLE OF CONTENTS, GLOSSARY, AND INDEX
To help you navigate through this guide, see the Contents, Glossary, and Index.

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

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

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

- The Electrostatics Interface, which simulates electric fields in dielectric materials with a fixed charge present. Preset stationary and time dependent study types are available.
• The Electric Currents Interface, which simulates the current in a conductive and capacitive material under the influence of an electric field. All three study types (stationary, frequency domain, and time dependent) are available.

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

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

• The Magnetic Fields Interface, which handles problems for magnetic fields with prescribed currents. All three study types (stationary, frequency domain, and time dependent) are available.

• The Magnetic Fields, No Currents Interface, which handles magnetic fields without currents. When no currents are present, the problem is easier to solve using the magnetic scalar potential. The stationary and time dependent study types are available.

• The Rotating Machinery, Magnetic Interface is available with 2D models only. It combines an out-of-plane magnetic fields (magnetic vector potential) formulation with a selection of predefined frames for prescribed rotation or rotation velocity—it shares most of its features with the Magnetic Fields interface. This interface requires that the geometry is created as an assembly from individual parts for the rotor and stator.

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

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

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

Heat transfer through conduction and convection in solids and free media (fluids) is supported by physics interfaces shipped with the basic COMSOL Multiphysics license. See also The Heat Transfer Interfaces, The Joule Heating Interface and Theory for the Heat Transfer Interfaces in the COMSOL Multiphysics User’s Guide.

MATERIALS

The Materials chapter has details about the electromagnetic material properties that you can store in the material databases such as electrical conductivity and resistivity, relative permittivity, relative permeability, nonlinear BH-curves, and refractive index.
This chapter contains an overview of the theory behind the AC/DC Module. It is intended for readers that wish to understand what goes on in the background when using the physics interfaces.

In this chapter:

- Fundamentals of Electromagnetics
- Electromagnetic Forces
- Special Calculations
- Electromagnetic Quantities
- References for the AC/DC Interfaces
Fundamentals of Electromagnetics

In this section:
- Maxwell’s Equations
- Constitutive Relations
- Potentials
- Reduced Potential PDE Formulations
- Electromagnetic Energy
- The Quasi-Static Approximation and the Lorentz Term
- Material Properties
- About the Boundary and Interface Conditions
- Phasors

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 \( \mathbf{E} \)
- Electric displacement or electric flux density \( \mathbf{D} \)
- Magnetic field intensity \( \mathbf{H} \)
- Magnetic flux density \( \mathbf{B} \)
- Current density \( \mathbf{J} \)
- Electric charge density \( \rho \)

The equations can be formulated in differential form or integral form. The differential form is presented here because it leads to differential equations that the finite element method can handle.

For general time-varying fields, Maxwell’s equations can be written as
The first two equations are also referred to as Maxwell-Ampère’s law and Faraday’s law, respectively. Equation three and four are two forms of Gauss’ law: the electric and magnetic form, respectively.

Another fundamental equation is the equation of continuity

\[ \nabla \cdot \mathbf{J} = \frac{\partial \rho}{\partial t} \]

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

**SEE ALSO**

- Constitutive Relations
- Potentials
- Reduced Potential PDE Formulations
- Electromagnetic Energy
- The Quasi-Static Approximation and the Lorentz Term
- Material Properties
- About the Boundary and Interface Conditions
- Phasors

**Constitutive Relations**

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

\[
\begin{align*}
\mathbf{D} &= \varepsilon_0 \mathbf{E} + \mathbf{P} \\
\mathbf{B} &= \mu_0 (\mathbf{H} + \mathbf{M}) \\
\mathbf{J} &= \sigma \mathbf{E}
\end{align*}
\]
where $\varepsilon_0$ is the **permittivity of vacuum**, $\mu_0$ is the **permeability of vacuum**, and $\sigma$ the **electrical conductivity**. In the SI system, the permeability of vacuum is chosen to be $4\pi \cdot 10^{-7}$ H/m. The velocity of an electromagnetic wave in vacuum is given as $c_0$ and the permittivity of vacuum is derived from the relation

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

The electromagnetic constants $\varepsilon_0$, $\mu_0$, and $c_0$ are available in COMSOL Multiphysics as predefined **physical constants**.

The **electric polarization vector** $\mathbf{P}$ describes how the material is polarized when an electric field $\mathbf{E}$ is present. It can be interpreted as the volume density of electric dipole moments. $\mathbf{P}$ is generally a function of $\mathbf{E}$. Some materials can have a nonzero $\mathbf{P}$ also when there is no electric field present.

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

For linear materials, the polarization is directly proportional to the electric field, $\mathbf{P} = \varepsilon_0 \chi_e \mathbf{E}$, where $\chi_e$ is the **electric susceptibility**. Similarly in linear materials, the magnetization is directly proportional to the magnetic field, $\mathbf{M} = \chi_m \mathbf{H}$, where $\chi_m$ 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 $\varepsilon_r$ is the **relative permittivity** and $\mu_r$ 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 $\varepsilon$ and $\mu$ (without subscripts) are the **permittivity** and **permeability** of the material.
GENERALIZED CONSTITUTIVE RELATIONS

Note: See also the Charge Conservation feature described for the Electrostatics interface (under the Electric Field section), which also describes the macroscopic properties of the medium (relating the electric displacement \( \mathbf{D} \) with the electric field \( \mathbf{E} \)) and the applicable material properties.

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

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

\[
\mathbf{B} = \mu_0 \mu_r \mathbf{H} + \mathbf{B}_r
\]

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

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

\[
\mathbf{B} = f(\|\mathbf{H}\|)
\]

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

SEE ALSO

- Maxwell’s Equations
- Potentials
- Reduced Potential PDE Formulations
- Electromagnetic Energy
- The Quasi-Static Approximation and the Lorentz Term
- Material Properties
- About the Boundary and Interface Conditions
- Phasors
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

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

The defining equation for the magnetic vector potential is a direct consequence of the magnetic Gauss’ law. The electric potential results from Faraday’s law.

In the magnetostatic case where there are no currents present, Maxwell-Ampère’s law reduces to $\nabla \times H = 0$. When this holds, it is also possible to define a magnetic scalar potential by the relation $H = -\nabla V_m$.

**SEE ALSO**
- Maxwell’s Equations
- Constitutive Relations
- Reduced Potential PDE Formulations
- Electromagnetic Energy
- The Quasi-Static Approximation and the Lorentz Term
- Material Properties
- About the Boundary and Interface Conditions
- Phasors

**Reduced Potential PDE Formulations**

The reduced potential option introduces the substitution $A = A_{\text{red}} + A_{\text{ext}}$ into Maxwell-Ampère’s law:

$$\nabla \times (\mu^{-1} \nabla \times A) = J + \frac{\partial D}{\partial t}$$

**DOMAIN EQUATIONS**

**Time-Harmonic**

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

$$\nabla \times \mathbf{A}_{\text{ext}} + \frac{1}{\sigma} \frac{\partial}{\partial t} \left( \mathbf{A}_{\text{ext}} + \mathbf{A}_{\text{red}} \right) = \mathbf{J}_e$$

### Static

In static formulations, the induced current is zero. Maxwell-Ampère’s law reduces to

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

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

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

### See Also
- Maxwell’s Equations
- Constitutive Relations
- Potentials
- Electromagnetic Energy
- The Quasi-Static Approximation and the Lorentz Term
- Material Properties
- About the Boundary and Interface Conditions
- Phasors

### Electromagnetic Energy

The electric and magnetic energies are defined as
\[
W_e = \int_V \left( \int_0^T \mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} \right) dV = \int_V \left( \int_0^T \mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} dt \right) dV
\]
\[
W_m = \int_V \left( \int_0^B \mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} \right) dV = \int_V \left( \int_0^T \mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} dt \right) dV
\]

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 \(S = \mathbf{E} \times \mathbf{H}\) is called the Poynting vector.

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

\[
\mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} = \varepsilon \mathbf{E} \cdot \frac{\partial \mathbf{E}}{\partial t} = \frac{\partial}{\partial t} \left( \frac{1}{2} \varepsilon \mathbf{E} \cdot \mathbf{E} \right)
\]
\[
\mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} = \frac{1}{\mu} \mathbf{B} \cdot \frac{\partial \mathbf{B}}{\partial t} = \frac{\partial}{\partial t} \left( \frac{1}{2} \frac{1}{\mu} \mathbf{B} \cdot \mathbf{B} \right)
\]
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), you get

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

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

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

**SEE ALSO**
- Maxwell’s Equations
- Constitutive Relations
- Potentials
- Reduced Potential PDE Formulations
- The Quasi-Static Approximation and the Lorentz Term
- Material Properties
- About the Boundary and Interface Conditions
- Phasors

**The Quasi-Static Approximation and the Lorentz Term**

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

The quasi-static approximation implies that the equation of continuity can be written as \( \nabla \cdot \mathbf{J} = 0 \) and that the time derivative of the electric displacement \( \varepsilon \mathbf{D}/\varepsilon t \) can be disregarded in Maxwell-Ampère’s law.
There are also effects of the motion of the geometries. Consider a geometry moving with velocity $v$ relative to the reference system. The force per unit charge, $F/q$, is then given by the Lorentz force equation:

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

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 $E' = E + v \times B$. In a conductive medium, the observer accordingly sees the current density

$$J = \sigma(E + v \times B) + J_e$$

where $J_e$ is an externally generated current density.

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

$$\nabla \times H = \sigma(E + v \times B) + J'$$

whereas Faraday’s law remains unchanged.

**See Also**

- Maxwell’s Equations
- Constitutive Relations
- Potentials
- Reduced Potential PDE Formulations
- Electromagnetic Energy
- Material Properties
- About the Boundary and Interface Conditions
- Phasors

**Material Properties**

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

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

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

**INHOMOGENEOUS MATERIALS**

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

**ANISOTROPIC MATERIALS**

For anisotropic materials the field relationships at any point differ for different directions of propagation. This means that a 3-by-3 tensor is necessary to properly define the constitutive relationships. If this tensor is symmetric, the material is often referred to as *reciprocal*. In such cases you can rotate the coordinate system such that a diagonal matrix results. If two of the diagonal entries are equal, the material is *uniaxially anisotropic*. If none of the elements has the same value, the material is *biaxially anisotropic* (Ref. 2). You need anisotropic parameters, for instance, 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.

**DISPERSE MATERIALS**

Dispersion describes changes in a wave’s velocity with wavelength. In the frequency domain you can express dispersion with a frequency dependence of the constitutive laws.

**SEE ALSO**

• Maxwell’s Equations
• Constitutive Relations
• Potentials
• Reduced Potential PDE Formulations
• Electromagnetic Energy
• The Quasi-Static Approximation and the Lorentz Term
• About the Boundary and Interface Conditions
• Phasors

About the Boundary and Interface Conditions

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

\[
\begin{align*}
\mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) &= 0 \\
\mathbf{n}_2 \cdot (\mathbf{D}_1 - \mathbf{D}_2) &= \rho_s \\
\mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) &= \mathbf{J}_s \\
\mathbf{n}_2 \cdot (\mathbf{B}_1 - \mathbf{B}_2) &= 0
\end{align*}
\]

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

From these relationships, you can derive the interface condition for the current density,

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

Interface Between a Dielectric and a Perfect Conductor

A perfect conductor has infinite electrical conductivity and thus no internal electric field. Otherwise, it would produce an infinite current density according to the third fundamental constitutive relation. At an interface between a dielectric and a perfect conductor, the boundary conditions for the \(\mathbf{E}\) and \(\mathbf{D}\) fields are simplified. Assume that subscript 1 corresponds to a perfect conductor; then \(\mathbf{D}_1 = 0\) and \(\mathbf{E}_1 = 0\) in the relationships just given. If, in addition, you are dealing with a time-varying case, then \(\mathbf{B}_1 = 0\) and \(\mathbf{H}_1 = 0\), 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:
- \mathbf{n}_2 \times \mathbf{E}_2 = 0  \\
- \mathbf{n}_2 \times \mathbf{H}_2 = \mathbf{J}_s  \\
- \mathbf{n}_2 \cdot \mathbf{D}_2 = \rho_s  \\
- \mathbf{n}_2 \cdot \mathbf{B}_2 = 0

**SEE ALSO**

- Maxwell’s Equations
- Constitutive Relations
- Potentials
- Reduced Potential PDE Formulations
- Electromagnetic Energy
- The Quasi-Static Approximation and the Lorentz Term
- Material Properties
- Phasors

**Phasors**

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

\[ E(\mathbf{r}, t) = \mathbf{E}(\mathbf{r}) \cos(\omega t + \phi) \]

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

\[ E(\mathbf{r}, t) = \mathbf{E}(\mathbf{r}) \cos(\omega t + \phi) = \text{Re}(\mathbf{\tilde{E}}(\mathbf{r})e^{i\phi}e^{i\omega t}) = \text{Re}(\mathbf{\tilde{E}}(\mathbf{r})e^{i\omega t}) \]

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

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

This means that an equation for the phasor can be derived from a time-dependent equation by replacing the time derivatives by a factor \( j\omega \). All time-harmonic equations in the AC/DC Module are expressed as equations for the phasors. (The tilde is dropped from the variable denoting the phasor.)
When analyzing the solution of a time-harmonic equation, it is important to remember that the field that has been calculated is a phasor and not a physical field. For example, all plot functions visualize $\text{Re}(E(r))$ by default, which is $E$ at time $t = 0$. To obtain the solution at a given time, you can specify a phase factor in all results pages and in the corresponding functions.

**SEE ALSO**

- Maxwell’s Equations
- Constitutive Relations
- Potentials
- Reduced Potential PDE Formulations
- Electromagnetic Energy
- The Quasi-Static Approximation and the Lorentz Term
- Material Properties
- About the Boundary and Interface Conditions
Electromagnetic Forces

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

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

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

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

Overview of Forces in Continuum Mechanics

Cauchy’s equation of continuum mechanics reads

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

where \( \rho \) is the density, \( \mathbf{r} \) denotes the coordinates of a material point, \( \mathbf{T} \) is the stress tensor, and \( \mathbf{f}_{\text{ext}} \) is an external volume force such as gravity (\( \mathbf{f}_{\text{ext}} = \rho \mathbf{g} \)). This is the equation solved in the structural mechanics physics interfaces for the special case of a linear elastic material, neglecting the electromagnetic contributions.
In the stationary case there is no acceleration, and the equation representing the force balance is

\[ 0 = \nabla \cdot T + f_{\text{ext}} \]

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,

\[ T = T_{\text{EM}} + \sigma_M \]

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 on the physics interfaces for structural mechanics, for example, Structural Mechanics Branch in the COMSOL Multiphysics User’s Guide.

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

\[ f_{\text{em}} = \nabla \cdot T_{\text{EM}} \]

This changes the force balance equation to

\[ 0 = \nabla \cdot \sigma_M + f_{\text{em}} + f_{\text{ext}} \]

or, as stated in the structural mechanics physics interfaces,
Forces on an Elastic Solid Surrounded by Vacuum or Air

Consider now 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. Note that 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 \( g_{\text{ext}} \). 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:

\[
\mathbf{n}_1 \left( T_2 - T_1 \right) = \mathbf{0} \\
\mathbf{n}_1 T_2 = \mathbf{n}_1 T_2 + g_{\text{ext}}
\]

The external boundary force \( g_{\text{ext}} \) 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

\[
\nabla \cdot T_1 + f_{\text{ext}} = \mathbf{0} \\
\mathbf{n}_1 (T_2 - T_1) + g_{\text{ext}} = \mathbf{0}
\]

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

\[
\int_{\Omega_1} (\nabla \cdot T_1 + f_{\text{ext}}) dV + \oint_{\partial \Omega_1} (\mathbf{n}_1 (T_2 - T_1) + g_{\text{ext}}) dS = \mathbf{0}
\]

Now, according to Gauss' theorem

\[
\int_{\Omega_1} \nabla \cdot T_1 dV - \oint_{\partial \Omega_1} \mathbf{n}_1 T_1 dS = \mathbf{0}
\]
This means that the external force

\[ \mathbf{F}_{\text{ext}} = \oint_{\Omega_i} \mathbf{f}_{\text{ext}} \, dV + \oint_{\partial \Omega_i} \mathbf{g}_{\text{ext}} \, dS \]

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

\[ \mathbf{F} = \oint_{\partial \Omega_i} \mathbf{n}_1 \mathbf{T}_2 \, dS \]

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

\[ m \mathbf{a} = \int_{\Omega_i} \rho \frac{d^2 \mathbf{r}}{dt^2} \, dV = \mathbf{F} \]

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. Note that to obtain this result, the contribution from the air pressure gradient has been neglected. This is equivalent to assuming that \( \nabla \cdot \mathbf{T}_2 = 0 \). A more detailed treatment shows that the pressure gradient contributes with a lifting (buoyancy) force on the solid.

**SEE ALSO**
- Overview of Forces in Continuum Mechanics
- Torque
- Forces in Stationary Fields
- Forces in a Moving Body
- Electromagnetic Energy and Virtual Work

**Torque**

The torque in the case of the previous section is given by

\[ \mathbf{M}_{\Omega} = \oint_{\partial \Omega_i} (\mathbf{r} - \mathbf{r}_\Omega) \times (\mathbf{n}_1 \mathbf{T}_2) \, dS \]
where \textbf{r}_O is a point on the axis of rotation. This follows from a derivation similar to the one made for forces.

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

**Forces in Stationary Fields**
The electromagnetic fields are stationary if

\[
\frac{\partial \mathbf{B}}{\partial t} = 0 \\
\frac{\partial \mathbf{D}}{\partial t} = 0
\]

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

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

**The Electromagnetic Stress Tensor**
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 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 $E$ and $B$ 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, $D = \varepsilon_0 E$ and $B = \mu_0 H$, the expression for the stress tensor can be written as

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

The equation for the balance of forces becomes

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

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

$$0 = -\nabla p + f_{\text{ext}}$$

Thus, using the same terminology as earlier, $f_{\text{em}} = 0$ for air, with $\sigma_M = -pI$. Note that 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

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

is useful (and similarly for $B$). 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,

$$n_1 T_2 = -p n_1 \left(\frac{1}{2} E \cdot D + \frac{1}{2} H \cdot B\right)n_1 + (n_1 \cdot E)D^T + (n_1 \cdot H)B^T$$

where $n_1$ 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 $F$ on the solid.
See the AC/DC Module Model Library model Permanent Magnet for an example of how to apply the stress tensor in air for calculating the total force and torque on a magnetizable rod close to a permanent magnet.

**Elastic Pure Conductor**

A material that is nonpolarizable and nonmagnetizable \((P = 0 \text{ and } M = 0)\) is called a *pure conductor*. Note that this is not necessarily equivalent to a perfect conductor, for which \(E = 0\), 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 \(-\rho I\) being replaced by the purely mechanical stress tensor \(\sigma_M\):

\[ T_1 = \sigma_M - \left( \frac{1}{2} E \cdot D + \frac{1}{2} H \cdot B \right) I + ED^T + HB^T \]

where \(D = \varepsilon_0 E\) and \(B = \mu_0 H\).

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

\[ J = \nabla \times H = \frac{1}{\mu_0} \nabla \times B \]

and the volume charge density

\[ \rho = \nabla \cdot D = \varepsilon_0 \nabla \cdot E \]

The equation for the balance of forces now becomes

\[ 0 = \nabla \cdot \sigma_M + \rho E + J \times B + f_{\text{ext}} \]

and this means that

\[ f_{\text{em}} = \rho E + J \times B \]

See the AC/DC Module Model Library model Electromagnetic Forces on Parallel Current-Carrying Wires 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.

**General Elastic Material**

For an elastic solid, in the general case of a material that is both dielectric and magnetic (nonzero \(P\) and \(M\)), the stress tensor is given by the expression
where in \( \sigma(E, B) \) the dependence of \( E \) and \( B \) has not been separated out. Thus \( \sigma \) is not a purely mechanical stress tensor in this general case. Different material models give different appearances of \( \sigma(E, B) \). The electromagnetic contributions to \( \sigma(E, B) \) 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 \( M = P = 0 \).

Note that \( T_1 \) must be symmetric. The terms \( EP^T \) and \( -MB^T \) are symmetric in the case of a linear dielectric and magnetic material because

\[
\sigma = \sigma_0 \varepsilon_0 \nabla \cdot E + \frac{1}{2\mu_0} \nabla \times B - M \cdot B \nabla
\]

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

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

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

**Forces in a Moving Body**

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

**FIELD TRANSFORMATIONS AND GALILEI INVARIANTS**

Assume that the object modeled is moving with a constant velocity, \( \mathbf{v} = \mathbf{v}_0 \). The equations now take on a slightly different form that includes the Galilei invariant versions of the electromagnetic fields. The term Galilei invariant is used due to the fact that they remain unchanged after a coordinate transformation of the type

\[
\mathbf{r}' = \mathbf{r} + \mathbf{v}_0 t
\]

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

The Galilei invariant fields of interest are

\[
\begin{align*}
\mathbf{E}' &= \mathbf{E} + \mathbf{v} \times \mathbf{B} \quad \text{(Electromotive intensity)} \\
\mathbf{J}' &= \mathbf{J} - \rho \mathbf{v} \quad \text{(Free conduction current density)} \\
\mathbf{P}' &= \mathbf{P} + \mathbf{v} (\nabla \cdot \mathbf{P}) - \nabla \times (\mathbf{v} \times \mathbf{P}) \quad \text{(Polarization flux derivative)} \\
\mathbf{M}' &= \mathbf{M} + \mathbf{v} \times \mathbf{P} \quad \text{(Lorentz magnetization)} \\
\mathbf{H}' &= \frac{\mathbf{B}}{\mu_0} - \epsilon_0 \mathbf{v} \times \mathbf{E} - \mathbf{M} \quad \text{(Magnetomotive intensity)}
\end{align*}
\]

As mentioned earlier 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 \( \epsilon_0 \mathbf{v} \times \mathbf{E} \) 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 you can neglect this term.

**Air and Vacuum**

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

\[
T_2 = -p I - \left( \frac{1}{2} \mathbf{E} \cdot \mathbf{D} + \frac{1}{2} \mathbf{H} \cdot \mathbf{B} \right) \mathbf{I} + \mathbf{E} \mathbf{D}^T + \mathbf{H} \mathbf{B}^T + (\mathbf{D} \times \mathbf{B}) \mathbf{v}^T
\]
Notice that there is an additional term in this expression compared to the stationary case.

**Elastic Pure Conductor**

The stress tensor in a moving elastic pure conductor is

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

where \(D = \varepsilon_0 E\) and \(B = \mu_0 H\).

To get the equation for the balance of forces you need to compute the divergence of this expression. Doing 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 reads

\[
\rho \frac{d^2 r}{dt^2} = \nabla \cdot T + f_{\text{ext}}
\]

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

\[
\rho \frac{d^2 r}{dt^2} = \nabla \cdot \sigma_M + \rho(\dot{E} + \dot{v} \times B) + \dot{\sigma}_M \times B + f_{\text{ext}}
\]

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

\[
0 = \nabla \cdot \sigma_M + \rho(\dot{E} + \dot{v} \times B) + (J - \tau \rho v) \times B + f_{\text{ext}}
\]

The terms containing \(\dot{v} \times B\) cancel out, which yields the following equation:

\[
0 = \nabla \cdot \sigma_M + \rho \dot{E} + J \times B + f_{\text{ext}}
\]

which is the same expression as for the stationary case.

**General Elastic Material**

The stress tensor for a moving general elastic material is
Notice that the magnetization $\mathbf{M}$ and the polarization $\mathbf{P}$ occur explicitly in this expression.

To instantiate the stress tensor for the general elastic case you need a material model explicitly including the magnetization and polarization effects as mentioned earlier in this section.

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

**Electromagnetic Energy and Virtual Work**

Another technique for calculating forces is the one of deriving the electromagnetic energy of the system and calculating the force by studying the effect of a small displacement. This is known as the method of virtual work or the principle of virtual displacement.

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

**Magnetic Force and Torque**

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

$$\mathbf{F}_\phi = -\nabla W_m$$

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

$$\mathbf{T}_\phi = \frac{\partial W_m}{\partial \phi}$$
where $\varphi$ is the rotational angle about the axis.

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

\[
F_I = \nabla W_m \\
T_I = \frac{\partial W_m}{\partial \varphi}
\]

**Electric Force and Torque**

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

\[
F_Q = -\nabla W_e \\
T_Q = \frac{\partial W_e}{\partial \varphi}
\]

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

\[
F_V = \nabla W_e \\
T_V = \frac{\partial W_e}{\partial \varphi}
\]

The method of virtual work can be employed by using the features for deformed mesh and sensitivity analysis in COMSOL Multiphysics. See Deformed Mesh Branch and Sensitivity Analysis in the COMSOL Multiphysics User’s Guide.

**See Also**

- 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
Special Calculations

In this section:
- Mapped Infinite Elements
- Lumped Parameter Conversion

Mapped Infinite Elements

In general, infinite elements are used at outer boundaries to model open boundaries, extending toward infinity. With proper settings infinite elements techniques enable termination of the simulations volume closer to the active regions (in other words, regions with sources), drastically reducing the amount of degrees of freedoms.

There are several different types of infinite elements, and the one used in the AC/DC Module is taken from Ref. 6. This technique is usually referred to as mapped infinite elements in the literature because it uses coordinate mapping of a region so its outer boundary is located at infinity. The principle can be explained in a one-coordinate system, where this coordinate represents Cartesian, cylindrical, or spherical coordinates. Mapping multiple coordinate directions (for Cartesian and cylindrical systems only) is just the sum of the individual coordinate mappings.

![Figure 2-1: The coordinate transform used for the mapped infinite element technique. The meaning of the different variables are explained in the text.](image)

Figure 2-1 shows a simple view of an arbitrary coordinate system. The coordinate \( r \) is the unscaled coordinate that COMSOL Multiphysics draw the geometry in (reference system). The position \( r_0 \) is the new origin from where the coordinates are scaled, \( t_p \) is the coordinate from this new origin to the beginning of the scaled region also called the pole distance, and \( \Delta_w \) is the unscaled length of the scaled region. The scaled coordinate, \( t' \), approaches infinity when \( t \) approaches \( t_p + \Delta_w \). To avoid solver issues...
with near infinite values, it is possible to change the infinite physical width of the scaled region to a finite large value, $\Delta_{pw}$. The true coordinate that the PDEs are formulated in is given by

$$r' = r_0 + t'$$

where $t'$ comes from the formula

$$t' = t_0 \frac{\Delta_w}{\Delta_p - \gamma(t - t_p)}$$

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

### Lumped Parameter Conversion

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

$$\mathbf{S} = G_{\text{ref}} \cdot (\mathbf{E} - (Z_{\text{ref}} \cdot Y)) \cdot (\mathbf{E} + Z_{\text{ref}} \cdot Y)^{-1} \cdot G_{\text{ref}}^{-1}$$

$$\mathbf{Z} = Y^{-1}, \quad \mathbf{L} = \frac{\text{Im}(\mathbf{Z})}{\omega}, \quad \mathbf{C} = \frac{\text{Im}(\mathbf{Y})}{\omega}, \quad \mathbf{R} = \text{Re}(\mathbf{Z}), \quad \mathbf{G} = \text{Re}(\mathbf{Y})$$

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

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

$$Z_{\text{ref}} = \mathbf{E} \cdot Z_0$$

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

where $Z_0$ is the characteristic impedance.
Electromagnetic Quantities

The table below shows the symbol and SI unit for most of the physical quantities that appear in the AC/DC Module. The default values for the permittivity of vacuum, $\varepsilon_0 = 8.854187817 \times 10^{-12} \text{ F/m}$, and for the permeability of vacuum, $\mu_0 = 4\pi \times 10^{-7} \text{ H/m}$, require that you provide all other quantities in SI units and that you use meter for the length scale of the geometry. If you draw the geometry using another length scale, it becomes necessary to change the numerical values for the physical quantities accordingly. For example, if you draw the geometry using $\mu \text{m}$ as the length scale, you need to have $\varepsilon_0 = 8.854187817 \times 10^{-18} \text{ F/\mu m}$ and $\mu_0 = 4\pi \times 10^{-13} \text{ H/\mu m}$.

<table>
<thead>
<tr>
<th>QUANTITY</th>
<th>SYMBOL</th>
<th>SI UNIT</th>
<th>ABBREVIATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angular frequency</td>
<td>$\omega$</td>
<td>radian/second</td>
<td>rad/s</td>
</tr>
<tr>
<td>Attenuation constant</td>
<td>$\alpha$</td>
<td>meter$^{-1}$</td>
<td>$m^{-1}$</td>
</tr>
<tr>
<td>Capacitance</td>
<td>$C$</td>
<td>farad</td>
<td>F</td>
</tr>
<tr>
<td>Charge</td>
<td>$q$</td>
<td>coulomb</td>
<td>C</td>
</tr>
<tr>
<td>Charge density (surface)</td>
<td>$\rho_s$</td>
<td>coulomb/meter$^2$</td>
<td>$C/m^2$</td>
</tr>
<tr>
<td>Charge density (volume)</td>
<td>$\rho$</td>
<td>coulomb/meter$^3$</td>
<td>$C/m^3$</td>
</tr>
<tr>
<td>Current</td>
<td>$I$</td>
<td>ampere</td>
<td>A</td>
</tr>
<tr>
<td>Current density (surface)</td>
<td>$J_s$</td>
<td>ampere/meter</td>
<td>A/m</td>
</tr>
<tr>
<td>Current density (volume)</td>
<td>$J$</td>
<td>ampere/meter$^2$</td>
<td>A/m$^2$</td>
</tr>
<tr>
<td>Electric displacement</td>
<td>$D$</td>
<td>coulomb/meter$^2$</td>
<td>$C/m^2$</td>
</tr>
<tr>
<td>Electric field</td>
<td>$E$</td>
<td>volt/meter</td>
<td>V/m</td>
</tr>
<tr>
<td>Electric potential</td>
<td>$V$</td>
<td>volt</td>
<td>V</td>
</tr>
<tr>
<td>Electric susceptibility</td>
<td>$\chi_e$</td>
<td>(dimensionless)</td>
<td>-</td>
</tr>
<tr>
<td>Electrical conductivity</td>
<td>$\sigma$</td>
<td>siemens/meter</td>
<td>S/m</td>
</tr>
<tr>
<td>Energy density</td>
<td>$W$</td>
<td>joule/meter$^3$</td>
<td>J/m$^3$</td>
</tr>
<tr>
<td>Force</td>
<td>$F$</td>
<td>newton</td>
<td>N</td>
</tr>
<tr>
<td>Frequency</td>
<td>$\nu$</td>
<td>hertz</td>
<td>Hz</td>
</tr>
<tr>
<td>Impedance</td>
<td>$Z$, $\eta$</td>
<td>ohm</td>
<td>$\Omega$</td>
</tr>
<tr>
<td>Inductance</td>
<td>$L$</td>
<td>henry</td>
<td>H</td>
</tr>
<tr>
<td>Magnetic field</td>
<td>$H$</td>
<td>ampere/meter</td>
<td>A/m</td>
</tr>
</tbody>
</table>
TABLE 2-1: ELECTROMAGNETIC QUANTITIES

<table>
<thead>
<tr>
<th>QUANTITY</th>
<th>SYMBOL</th>
<th>SI UNIT</th>
<th>ABBREVIATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Magnetic flux</td>
<td>( \Phi )</td>
<td>weber</td>
<td>Wb</td>
</tr>
<tr>
<td>Magnetic flux density</td>
<td>( \mathbf{B} )</td>
<td>tesla</td>
<td>T</td>
</tr>
<tr>
<td>Magnetic potential (scalar)</td>
<td>( V_m )</td>
<td>ampere</td>
<td>A</td>
</tr>
<tr>
<td>Magnetic potential (vector)</td>
<td>( \mathbf{A} )</td>
<td>weber/meter</td>
<td>Wb/m</td>
</tr>
<tr>
<td>Magnetic susceptibility</td>
<td>( \chi_m )</td>
<td>(dimensionless)</td>
<td>-</td>
</tr>
<tr>
<td>Magnetization</td>
<td>( \mathbf{M} )</td>
<td>ampere/meter</td>
<td>A/m</td>
</tr>
<tr>
<td>Permeability</td>
<td>( \mu )</td>
<td>henry/meter</td>
<td>H/m</td>
</tr>
<tr>
<td>Permittivity</td>
<td>( \varepsilon )</td>
<td>farad/meter</td>
<td>F/m</td>
</tr>
<tr>
<td>Polarization</td>
<td>( \mathbf{P} )</td>
<td>coulomb/meter(^2)</td>
<td>C/m(^2)</td>
</tr>
<tr>
<td>Poynting vector</td>
<td>( \mathbf{S} )</td>
<td>watt/meter(^2)</td>
<td>W/m(^2)</td>
</tr>
<tr>
<td>Propagation constant</td>
<td>( \beta )</td>
<td>radian/meter</td>
<td>rad/m</td>
</tr>
<tr>
<td>Reactance</td>
<td>( X )</td>
<td>ohm</td>
<td>( \Omega )</td>
</tr>
<tr>
<td>Relative permeability</td>
<td>( \mu_r )</td>
<td>(dimensionless)</td>
<td>-</td>
</tr>
<tr>
<td>Relative permittivity</td>
<td>( \varepsilon_r )</td>
<td>(dimensionless)</td>
<td>-</td>
</tr>
<tr>
<td>Resistance</td>
<td>( R )</td>
<td>ohm</td>
<td>W</td>
</tr>
<tr>
<td>Resistive loss</td>
<td>( Q )</td>
<td>watt/meter(^3)</td>
<td>W/m(^3)</td>
</tr>
<tr>
<td>Torque</td>
<td>( T )</td>
<td>newton-meter</td>
<td>N·m</td>
</tr>
<tr>
<td>Velocity</td>
<td>( \mathbf{v} )</td>
<td>meter/second</td>
<td>m/s</td>
</tr>
<tr>
<td>Wavelength</td>
<td>( \lambda )</td>
<td>meter</td>
<td>m</td>
</tr>
<tr>
<td>Wave number</td>
<td>( k )</td>
<td>radian/meter</td>
<td>rad/m</td>
</tr>
</tbody>
</table>
References for the AC/DC Interfaces


Modeling with the AC/DC Module

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

In this chapter:

- Preparing for Modeling
- Infinite Elements
- Force and Torque Computations
- Lumped Parameters
- Lumped Ports with Voltage Input
- S-Parameters and Ports
- Importing ECAD Files
Preparing for Modeling

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

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

See also **Overview of the Physics Interfaces** in the COMSOL Multiphysics User’s Guide for general guidelines for effective modeling.

**GENERAL TIPS**

These general tips about modeling will help you to decide what to include in your simulation and what you can do to minimize the size of your problem. Before you start modeling, try first to answer the following questions:

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

It is important to remember that a model never captures 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. Keep in mind that 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 handles static, time-dependent, and time-harmonic problems. The time-dependent and time-harmonic formulations use a quasi-static approximation. See Table 1-1 in the section Overview of the User’s Guide for a list of the preset study types available by interface.

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

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

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

Independently 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 you can 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 models that comes with this product.

**Selecting the Space Dimension for the Model Geometry**

Most of the problems that you solve 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 to or equivalent to your real problem. Furthermore, it is good practice to start a modeling project by building one or several 2D models before going to a 3D model. This is because 2D models are easier to modify and solve much faster. Thus, modeling mistakes are much easier to find when...
working in 2D. Once you have verified your 2D model, you are in a much better position to build a 3D model.

**2D PROBLEMS**

The following guides you through some of the common approximations made for 2D problems. Remember that the 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 you know there is no variation of the solution in one particular dimension
- When you have a problem where you can neglect the influence of the finite extension in the third dimension

See the AC/DC Module Model Library model Electromagnetic Forces on Parallel Current-Carrying Wires for an example. The geometry has a finite width but the model neglects the (end) effects from the faces parallel to the cross section because the strongest forces are between the perpendicular faces (those seen as lines in the cross section).

![Figure 3-1: The cross sections and their real geometry for Cartesian coordinates and cylindrical coordinates (axial symmetry).](image-url)
Axial Symmetry (Cylindrical Coordinates)

If you can construct the 3D geometry by revolving a cross section about an axis, and no variations in any variable occur when going around the axis of revolution, you can use an axisymmetric physics interface. The spatial coordinates are called \( r \) and \( z \), where \( r \) is the radius. The flow at the boundaries is given per unit length along the third dimension. Because this dimension is a revolution, you have to multiply all flows with \( \alpha r \), where \( \alpha \) is the revolution angle (for example, \( 2\pi \) for a full turn).

3D Problems

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 you spend on simplifying your problem is probably well spent when solving it. Below are a few issues that should be addressed before starting to implement a 3D model.

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

Are there symmetries in the geometry and model? Many problems have planes where the solution on either side of the plane looks the same. A good way to check this is to flip the geometry around the plane, for example, by turning it up-side 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. See the AC/DC Module Model Library model Eddy Currents for an example.

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 the next topic Simplifying the Geometry Using Boundary Conditions).

See Also

- What Problems Can You Solve?
- Simplifying the Geometry Using Boundary Conditions
- Applying Electromagnetic Sources
- Selecting a Study Type
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 suggestions of how to do this in both 2D and 3D problems.

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

**Can you replace the thin layers with boundary conditions?** There are several types of boundary conditions in COMSOL Multiphysics suitable for such replacements. You can, for example, replace materials with high conductivity with the shielding boundary condition, which assumes a constant potential through the thickness of the layer. If you have a magnetic material with a high relative permeability, you can also model it using the shielding boundary condition. See the AC/DC Module Model Library One-Sided Magnet and Plate.

**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. See the AC/DC Module Model Library Cold Crucible.

**SEE ALSO**
- What Problems Can You Solve?
- Selecting the Space Dimension for the Model Geometry
- Applying Electromagnetic Sources
- Selecting a Study Type
- Field Variables in 2D
- Meshing and Solving
**Applying Electromagnetic Sources**

You can apply electromagnetic sources 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 you can compute 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.

**SEE ALSO**
- What Problems Can You Solve?
- Selecting the Space Dimension for the Model Geometry
- Simplifying the Geometry Using Boundary Conditions
- Selecting a Study Type
- Field Variables in 2D
- Meshing and Solving

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

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

You need to 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 you have a periodic input with many harmonics, like a square-shaped signal.

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

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

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

**SEE ALSO**
- What Problems Can You Solve?
- Selecting the Space Dimension for the Model Geometry
- Simplifying the Geometry Using Boundary Conditions
- Applying Electromagnetic Sources
- Field Variables in 2D
- Meshing and Solving

*Field Variables in 2D*

When you want to solve for a vector field in 2D, the physics interface gives you three options: you can solve for the out-of-plane vector, the in-plane vector, or the three-component vector. Depending on what you choose, the available source specification options on the domain, boundary, edge, and point levels change accordingly.
SEE ALSO

- What Problems Can You Solve?
- Selecting the Space Dimension for the Model Geometry
- Simplifying the Geometry Using Boundary Conditions
- Applying Electromagnetic Sources
- Selecting a Study Type
- Meshing and Solving

Meshing and Solving

MESH RESOLUTION

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

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

Is the skin effect or the field variation due to losses? It is easy to estimate the skin depth from the conductivity, permeability, and frequency. You need at least two linear elements per skin depth to capture the variation of the fields. If you do not study the skin depth, you can replace regions with a small skin depth with a boundary condition, thereby saving elements.

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

SELECTING A SOLVER

You can, in most cases, use the solver that COMSOL Multiphysics suggests. 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 you might need to tune the solver settings. This is especially important for 3D problems because they use a large...
amount of memory. For large 3D problems, you may need a 64-bit platform. See Solvers and Study Types in the COMSOL Multiphysics User’s Guide for a more detailed description.

**SEE ALSO**
- What Problems Can You Solve?
- Selecting the Space Dimension for the Model Geometry
- Simplifying the Geometry Using Boundary Conditions
- Applying Electromagnetic Sources
- Selecting a Study Type
- Field Variables in 2D
Infinite Elements

In this section:
- Modeling Unbounded Domains
- Known Issues When Modeling Using Infinite Elements

Modeling Unbounded Domains

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

Another approach is to use infinite elements. There are many implementations of infinite elements available, and the elements used in this Module are often referred to as mapped infinite elements (see Ref. 1). This implementation maps the model coordinates from the local, finite-sized domain to a stretched domain. The inner boundary of this stretched domain coincides with the local domain, but at the exterior boundary the coordinates are scaled toward infinity:

\[
\begin{align*}
  t' &= t_p \frac{\Delta_w}{\Delta_{pw} - \gamma(t - t_0)} \\
  \gamma &= 1 - \frac{t_p}{\Delta_{pw} + t_p}
\end{align*}
\]

The pole distance, \(t_p\), and the physical width of the infinite element region, \(\Delta_{pw}\), are input parameters for the region. The variable \(t\) is the unscaled coordinate along the width of the infinite element region (from inner to outer boundary), \(t_0\) is the start position for the region, and \(\Delta_w\) is the unscaled width of the region. The software automatically computes the value for this variable and the orientation of the transform for infinite element regions that are Cartesian, cylindrical, or spherical. However, there is no check that the geometry of the region is correct, so it is important to draw a proper geometry and select the corresponding region type.
The following figures show typical examples of infinite element regions that work nicely for each of the infinite element types. These types are:

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

*Figure 3-2: A cube surrounded by typical infinite-element regions of Cartesian type.*

*Figure 3-3: A cylinder surrounded by typical cylindrical infinite-element regions. Cylindrical infinite elements are only supported in 2D axisymmetry.*
Figure 3-4: A sphere surrounded by a typical spherical infinite-element region.

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

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

**General Stretching**

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

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

When going from any of the other types to the General type, subnodes that represent stretching of the previous type are added automatically.
**Known Issues When Modeling Using Infinite Elements**

Be aware of the following when modeling with infinite elements:

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

**Element Quality**
The coordinate scaling resulting from infinite elements also yields an equivalent stretching or scaling of the mesh that effectively results in a poor element quality. (The element quality displayed by the mesh statistics feature does not account for this effect.)

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

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

**Erroneous Results**
Infinite element regions deviating significantly from the typical configurations shown in the beginning of this section can cause the automatic calculation of the infinite element parameter to give erroneous result. Enter the parameter values manually if you find that this is the case. See General Stretching.
Use the Same Material Parameters or Boundary Conditions
The infinite element region is designed to model uniform regions extended toward infinity. Avoid using objects with different material parameters or boundary conditions that influence the solution inside an infinite element region.

See Also
• Modeling Unbounded Domains

Reference for Infinite Elements
Force and Torque Computations

In this section:
- Calculating Electromagnetic Forces and Torques
- Model Examples—Electromagnetic Forces

Calculating Electromagnetic Forces and Torques

To calculate electromagnetic forces and torques in the AC/DC Module two methods are available:
- The most general method is to use the Maxwell stress tensor.
- Another method that works for the special case of computation of magnetic forces on nonmagnetic, current-carrying domains uses a predefined physics interface variable for the Lorentz force distribution in a magnetic flux density $B$.

Maxwell Stress Tensor

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

$$n_1 T_2 = -\frac{1}{2} n_1 (E \cdot D) + (n_1 \cdot E) D^T$$

(3-1)

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

In the magnetic fields interface, the expression

$$n_1 T_2 = -\frac{1}{2} n_1 (H \cdot B) + (n_1 \cdot H) B^T$$

is integrated on the surface to obtain the force. In the magnetic and electric fields interface, both expressions are included. $E$ is the electric field, $D$ the electric displacement, $H$ the magnetic field, $B$ the magnetic flux density, and $n_1$ the outward normal from the object.

For a theoretical discussion about the stress tensor see Electromagnetic Forces.
LORENTZ FORCES

The Lorentz force is defined as \( \mathbf{F} = \mathbf{J} \times \mathbf{B} \). The Lorentz force is very accurate for electromagnetic force calculations in electrically conducting domains. The Lorentz force variables are available both in domains and on boundaries (in the case of surface currents).

Model Examples—Electromagnetic Forces

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

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

The Permanent Magnet model demonstrates how to compute the total force on a magnetizable rod close to a permanent magnet by integrating the Maxwell stress tensor in the air on the outside of the rod. This is the most important method for accurately calculating the total force on magnetic devices for which the exact distribution of volume forces is not known. To retrieve the exact distribution of volume forces requires a material model that describes the interactions of the magnetizations and strains. Such material models are not always available. Therefore you are often limited to compute the total force by integrating the stress tensor or using the method of virtual work. Note that you cannot use any of these methods to compute and visualize the force distribution inside a domain, only to compute the total force and torque in situations where the device is surrounded by air (or when this is a good approximation).
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 you only get the resistive, capacitive, or inductive part of the lumped parameter matrix.

In this section:
- Calculating Lumped Parameters with Ohm’s Law
- Calculating Lumped Parameters Using the Energy Method
- Studying Lumped Parameters

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. You can force either a voltage or a current on the electrodes. After the simulation you can extract the other property or you can extract 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 interface that you use, what parameter you are interested in, and how you solve the model. The overview of the techniques below uses a 4-by-4 matrix example for the lumped parameter matrix. This represents a system of at least five terminals, where four are used as terminals and the rest are grounded, as illustrated in Figure 3-5.

![Figure 3-5: A five-terminal system with 4 terminals and one grounded terminal.](image)

**Figure 3-5:** A five-terminal system with 4 terminals and one grounded terminal.
If you specify a system where all terminals are terminals, you get redundant matrix elements. This is better understood if you view a two-terminal system. If both terminals are declared as terminals, you get a 2-by-2 matrix for the system. This is clearly too many elements because you only have one unique lumped parameter between the terminals. As soon as you declare other ground terminals somewhere in the system, you get a 3-terminal system 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

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

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

\[
\begin{bmatrix}
Q_1 \\
Q_2 \\
Q_3 \\
Q_4
\end{bmatrix} =
\begin{bmatrix}
C_{11} & C_{12} & C_{13} & C_{14} \\
C_{21} & C_{22} & C_{23} & C_{24} \\
C_{31} & C_{32} & C_{33} & C_{34} \\
C_{41} & C_{42} & C_{43} & C_{44}
\end{bmatrix}
\begin{bmatrix}
V_1 \\
V_2 \\
V_3 \\
V_4
\end{bmatrix}
\]

**FIXED CURRENT**

It might be necessary to calculate the \( \mathbf{Z} \)-matrix in a more direct way. Similar to the \( \mathbf{Y} \) calculation, the \( \mathbf{Z} \) 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:
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 you the inverted capacitance matrix in a similar manner as the \( Z \) and \( Y \) matrices.

\[
\begin{bmatrix}
V_1 \\
V_2 \\
V_3 \\
V_4
\end{bmatrix} = \begin{bmatrix}
Z_{11} & Z_{12} & Z_{13} & Z_{14} \\
Z_{21} & Z_{22} & Z_{23} & Z_{24} \\
Z_{31} & Z_{32} & Z_{33} & Z_{34} \\
Z_{41} & Z_{42} & Z_{43} & Z_{44}
\end{bmatrix}^{-1} \begin{bmatrix}
I_1 \\
I_2 \\
I_3 \\
I_4
\end{bmatrix}
\]

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

**SEE ALSO**

- Calculating Lumped Parameters Using the Energy Method
- Studying Lumped Parameters

*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 you extract the energy density integrated over the whole geometry. The following formulas show how to calculate the capacitance matrix from the integral of the electric energy density.

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

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

Calculate the inductance matrix in the same way from the magnetic energy density:
This is the technique used when Fixed current is selected.

**SEE ALSO**
- Calculating Lumped Parameters with Ohm’s Law
- Studying Lumped Parameters

**Studying Lumped Parameters**

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

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

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

**PORT SWEEP SETTINGS AND TOUCHSTONE EXPORT**

In the main node of the interface, activate a port 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 port/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 you 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 you cannot verify this without using reaction terms.

SEE ALSO

- Calculating Lumped Parameters with Ohm’s Law
- Calculating Lumped Parameters Using the Energy Method
Lumped Ports with Voltage Input

In this section:

- About Lumped Ports
- Lumped Port Parameters

About Lumped Ports

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

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

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

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

where $V_{\text{port}}$ is the extracted voltage for the port given by the line integral between the terminals averaged over the entire port. The current $I_{\text{port}}$ 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. For more details, see also Lumped Port Parameters.

Lumped Port Parameters

In transmission line theory you deal with voltages and currents 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

\[ V = \int_{h} E \cdot dl = \int_{h} (E \cdot a_h) dl \]

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

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

The impedance of a transmission line is defined as

\[ Z = \frac{V}{I} \]

and in analogy to this you can define an equivalent surface impedance at the lumped port boundary

\[ \eta = \frac{E \cdot a_h}{J_s \cdot (-a_h)} \]

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

\[ I = \int_{w} (n \times J_s) \cdot dl = -\int_{w} (J_s \cdot a_h) dl \]

where the integration is taken in the direction of \( a_h \times n \). This gives the following relation between the transmission line impedance and the surface impedance.
where the last approximation assumed that the electric field is constant over the integrations. A similar relationship can be derived for coaxial cables

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

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

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

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

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

In this section:

- S-Parameters in Terms of Electric Field
- S-Parameter Calculations in COMSOL Multiphysics: Lumped Ports
- S-Parameter Variables

S-Parameters in Terms of Electric Field

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

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

\[
S = \begin{bmatrix}
S_{11} & S_{12} & \cdots & S_{1n} \\
S_{21} & S_{22} & \cdots & \cdots \\
\vdots & \vdots & \ddots & \vdots \\
S_{n1} & \cdots & \cdots & S_{nn}
\end{bmatrix}
\]

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

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

S-Parameter Calculations in COMSOL Multiphysics: Lumped Ports

The AC/DC interfaces have a built-in support for S-parameter calculations. To set up an S-parameter study use a Lumped Port boundary feature for each port in the model.
The lumped ports should only be used when the port width is much smaller than the wavelength. For more details about lumped ports, see Lumped Ports with Voltage Input. See Lumped Port for instructions to set up your model.

**S-Parameter Variables**

The AC/DC Module automatically generates variables for the S-parameters. The port names (use numbers for port sweeps to work correctly) determine the variable names. If you, for example, have two lumped ports with the numbers 1 and 2 and Lumped Port 1 is the inport, the software generates the variables $S_{11}$ and $S_{21}$. $S_{11}$ is the S-parameter for the reflected wave and $S_{21}$ is the S-parameter for the transmitted wave. For convenience, two variables for the S-parameters on a dB scale, $S_{11\text{dB}}$ and $S_{21\text{dB}}$, 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 may vary. The S-parameter variables are added to the predefined quantities in appropriate plot lists.
Importing ECAD Files

In this section:

- Overview of the ECAD Import
- Importing ODB++(X) Files
- Importing GDS-II Files
- Importing NETEX-G Files
- ECAD Import Options
- Meshing an Imported Geometry
- Troubleshooting ECAD Import

Overview of the ECAD Import

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

Extruding Layers

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

As a result of these performance issues, the ECAD Import has its own extrude operation that automatically connects non matching planes. In one operation this functionality extrudes and connects all layers, so there is only one geometry object after the import. With only one object, it is easy to switch to the physics modes. You use this special extrude operation when you use the grouping option All.
The special extrude operation is bound to certain rules that the 2D layout must fulfill. If the 2D layout does not comply with these rules, the operation might fail. You can then switch to one of the other grouping options to import the geometry.

**SEE ALSO**
- Importing ODB++(X) Files
- Importing GDS-II Files
- Importing NETEX-G Files
- ECAD Import Options
- Meshing an Imported Geometry
- Troubleshooting ECAD Import

**Importing ODB++(X) Files**

**Note:** If your ECAD software supports the ODB++(X) format it is recommended that you use it as it usually gives the most efficient geometry model of the layout.

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

ODB++ exists in two different format versions:

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

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

**Extracting Layer Stackup**
The import can read stackup information from the ODB++ file, such as thickness for metal layers and dielectric layers. It is quite common that the layer thickness is not
included in the export from the ECAD program, so the layers only get a default thickness. You always have the possibility to change the thickness prior to import on the Layers to import table in the Settings window for the ECAD import, so it is recommended that you check these values before importing.

SEE ALSO

- Overview of the ECAD Import
- Importing GDS-II Files
- Importing NETEX-G Files
- ECAD Import Options
- Meshing an Imported Geometry
- Troubleshooting ECAD Import

Importing GDS-II Files

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

File Extension

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

SUPPORTED FEATURES

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

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

3D IMPORT OF GDS-II FILES

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

• One layer represents one position in height, so if the file contains two GDS layers that define two objects on the same height, the ECAD import still positions the layers with one layer on top of the other. Several GDS layers on the same height is common for semiconductor layouts, where the fabrication process includes deposition followed by etching and then redepositing of a different layer. Such advanced process schemes cannot be automatically handled correctly by the ECAD import.

• With the grouping option All, objects on adjacent layers must not cross each other, because the original edge of the objects must be kept unchanged when two adjacent layers are merged to form the interface between them. You can get around this by selecting a different grouping option (see ECAD Import).

• Use the 3D GDS-II import with the ECAD import. The standard CAD import of COMSOL Multiphysics does not support pre-reading of the file, so it is not possible to specify any properties the layers (like thickness for example). The ECAD import always reads the file before displaying the import options.

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

SEE ALSO
• Overview of the ECAD Import
Importing NETEX-G Files

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

**File Extension**

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

**USING NETEX-G**

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

**GERBER Layer Files**

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

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

Because the Gerber layer files do not contain any physical information about the layer and dielectrics, you also need to specify this information in NETEX-G.

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

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

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

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

**Types of Wirebonds**
The ECAD import can model the wirebond at three different complexity levels:

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

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

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

**SEE ALSO**
- Overview of the ECAD Import
- Importing ODB++(X) Files
- Importing GDS-II Files
- ECAD Import Options
- Meshing an Imported Geometry
- Troubleshooting ECAD Import

**ECAD Import Options**

**ECAD IMPORT**

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

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

Before clicking the Import button you should consider the import options described below.
THE ECAD IMPORT OPTIONS

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

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

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

- **All.** Groups all objects into one single object. This selection makes use of a more efficient extrude algorithm that extrudes and combines all layers directly. Because the import results in only one geometry object, COMSOL Multiphysics does not need to do a complicated analysis of several geometry objects.

- **By layer.** Groups all objects in one layer into one geometry object. The final geometry contains one object for each layer.

- **No grouping.** No grouping of objects is performed. This can be useful for debugging purposes when the other choices fail for some reason. This selection returns all the primitive objects found in the file, so objects with negative polarity are not drawn correctly.

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

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

- **Edges.** The path of the bond wire is represented only as a geometrical edge. This option has the least complexity and does not produce a large number of mesh elements. There might be some limitations when using these edges in modeling.

- **Blocks.** The bond wire is modeled as a solid with a square cross section.

- **Cylinders.** Same as above but with a circular cross section.

Select the Manual control of elevations check box to manually position the layers in the z direction. This check box is enabled when Grouping of geometries is set to By layer or No grouping. When Manual control of elevations is not enabled, the z positions of the layers are calculated automatically from the layer Thickness values.
The layer information from the file appears in the **Layers to import** table. In addition to the layer **Name**, the table includes the following columns:

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

- **The numbers in the Thickness column can be changed.** This column is especially important when importing GDS files because that format does not contain any thickness information, so all layers get a default thickness that you probably want to change.

- **The number in the Elevation column can be changed.** The Elevation column controls the lower Z position of a layer. The Elevation column is only displayed when **Manual control of elevations** is enabled.

- **The Import column.** Here you can clear the check box for layers that you do not want to import. Note that if you use the **Metal shells** import type, you cannot import isolated boundaries if the import also includes another solid layer. Then you must perform two imports. The only exception to this rule is when the import results in only face objects.

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

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

Clearing the **Ignore text objects** check box tells the importer to skip all objects in an ODB++ file that have the **TEXT** tag set. It is common that PCB layouts have text written in copper. Such objects increase the problem size and are usually of no interest in a physical simulation.
For NETEX-G/GDS import, other options that can significantly reduce the complexity of imported layouts are the recognition of arcs and straight lines. With the recognize arcs set to Automatic, all polygon chains that represent arcs are identified and replaced with more efficient curve objects. With the edit fields appearing when setting this to Manual, you can fine tune the arc recognition. The find straight lines check box also controls whether to convert several polygon segments that lie on a single straight line into a single straight segment. This option uses the number in the minimum angle between segments edit field to determine if a group of segments lies on the same straight line.

Geometry repair is controlled via the repair imported data check box and the relative repair tolerance edit field.

See Also
- Overview of the ECAD Import
- Importing ODB++(X) Files
- Importing GDS-II Files
- Importing NETEX-G Files
- Meshing an Imported Geometry
- Troubleshooting ECAD Import

Meshing an Imported Geometry

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

The following section describes this procedure in general terms.

This procedure assumes that the top and bottom layers are metal layers. All metal layers can often be meshed using swept meshing, but dielectric layers usually cannot be meshed that way. You begin by meshing from the bottom or top layer, starting with a boundary mesh. Then you mesh layer by layer, where each metal layer gets a swept mesh, and each dielectric layer (with vias) gets a free mesh.

The dielectric layers cannot use a swept mesh because the source and target boundaries usually do not look the same. If there is a surrounding air domain it is usually not possible to use swept meshes for the metal layers either. You must then use tetrahedrons or convert the swept mesh to tetrahedrons before meshing the surrounding domain. For more details see Creating Meshes and Generating a 3D...

SEE ALSO
• Overview of the ECAD Import
• Importing ODB++(X) Files
• Importing GDS-II Files
• Importing NETEX-G Files
• ECAD Import Options
• Troubleshooting ECAD Import

Troubleshooting ECAD Import

TUNING IMPORT SETTINGS

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

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

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

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

**PROBLEMS WITH SEVERAL GEOMETRY OBJECTS**

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

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

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

**SEE ALSO**

- Overview of the ECAD Import
- Importing ODB++(X) Files
- Importing GDS-II Files
- Importing NETEX-G Files
- ECAD Import Options
- Meshing an Imported Geometry
The Electric Field Interfaces

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

In this chapter:

- The Electrostatics Interface
- The Electric Currents Interface
- The Electric Currents, Shell Interface
- Theory of Electric Fields
- Theory for the Electrostatics Interface
- Theory for the Electric Currents Interface
- Theory for the Electric Currents, Shell Interface
The Electrostatics Interface

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

For an introduction to the physics and equations implemented by this interface, see the Theory for the Electrostatics Interface.

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

When you add this interface, these default nodes are also added to the Model Builder—Charge Conservation, Zero Charge (default boundary condition), and Initial Values.

Right-click the Electrostatics node to add other features that implement, for example, boundary conditions and space charges.

To display additional features for the physics interfaces and feature nodes, click the Show button in the Model Builder and select the applicable section.

SHOW MORE OPTIONS FOR PHYSICS INTERFACES AND FEATURE NODES

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

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

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

Availability of each feature, and whether it is described for a particular interface or node, is based on the individual physics interface and feature node. For example, the Discretization, Advanced Settings, Consistent Stabilization, and Inconsistent Stabilization sections are often described individually throughout the documentation as there are unique settings. See Showing and Expanding Advanced Feature Nodes and Sections.
in the *COMSOL Multiphysics User’s Guide* for additional links to the relevant documentation.

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

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

**DOMAIN SELECTION**
Select the domains where you want to define the electric potential and the equations that describe the potential field for dielectrics. The default setting is to include all domains in the model.

**OUT-OF-PLANE THICKNESS (2D MODELS ONLY)**
Define the out-of-plane thickness $d$ by entering a value or expression (SI unit: m) in the Thickness edit field. 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 models.

**PORT SWEEP SETTINGS**
Select the Activate port sweep check box to switch on the port sweep and invoke a parametric sweep over the ports/terminals. Enter a Port parameter name to assign a specific name to the variable that controls the port or terminal number solved for during the sweep. The generated lumped parameters are in the form of capacitance matrix elements. The port/terminal settings must consistently be of either fixed voltage or fixed charge type. See Lumped Parameters for more information.

**Solving for a Port Sweep**
An additional step is required when solving for a port sweep. You need to right-click the Study node in the model tree and add a Parametric Sweep. In the new Parametric Sweep node, you enter for Parameter names, the Port parameter name specified when activating the port sweep and for Parameter values, you enter the desired list with terminal numbers. You then need to right-click the Study node and select Other and Generate Sequences from Study before solving.
DISCRETIZATION
To display this section, select click the Show button ( ) and select Discretization.
Select an element order for the Electric potential—Linear, Quadratic (the default), Cubic, Quartic, or (in 2D only) Quintic.

DEPENDENT VARIABLES
The dependent variable (field variable) is for the Electric potential \( V \). You can change the name in the corresponding edit field, but the names of fields and dependent variables must be unique within a model.

SEE ALSO
- Charge Conservation
- Space Charge Density
- Force Calculation
- Infinite Elements
- Initial Values
- Boundary Conditions for the Electrostatics Interface
- Pairs for the Electrostatics Interface
- Line Charge
- Point Charge
- Electrostatic Point Dipole

Charge Conservation

The Charge Conservation node adds the equations for charge conservation according to Gauss’ law for the electric displacement field. The Charge Conservation page contains these sections for defining the related material properties:

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show
button (Button) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Domain Selection**
Select the domains where you want to define the electric potential and the equation based on Gauss’ law that describes the potential field.

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

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

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

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

- If Relative permittivity is selected, the default is to take the Relative permittivity ($\varepsilon_r$) values From material. If User defined is selected, select Isotropic, Diagonal, Symmetric, or Anisotropic and enter values or expressions in the field or matrix.
- If Polarization is selected, enter components (3 in 3D, 2 in 2D) for the Polarization vector $\mathbf{P}$ (SI unit: $C/m^2$).
- If Remanent displacement is selected, the default is to take the Relative permittivity ($\varepsilon_r$) values From material. If User defined is selected, select Isotropic, Diagonal, Symmetric, or Anisotropic and enter values or expressions in the field or matrix. Then enter components (3 in 3D, 2 in 2D) for the Remanent displacement $\mathbf{D}_r$ (SI unit: $C/m^2$).
Space Charge Density

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

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (-show) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

DOMAIN SELECTION

Select the domains where you want to define a current source.

SPACE CHARGE DENSITY

Enter a value or expression for the Space charge density $p$ (SI unit: C/m$^3$).

Force Calculation

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

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show
button (†) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**DOMAIN SELECTION**

Select the domains where you want to define a force calculation.

**FORCE CALCULATION**

Enter a Force name, which is then available as a global variable. The method used is integration of the Maxwell’s stress tensor over the exterior surfaces of the set of domains. This feature also gives access to the normal component of the Maxwell Stress tensor on the external surfaces. (For the Magnetic and Electric Fields interface, the force calculation includes both electric and magnetic forces).

Enter coordinates for the Torque axis \( r_{ax} \) and Torque rotation point \( r_0 \). A torque calculation about a given point (Torque rotation point) is made, and the resulting torque component parallel to the given Torque axis is given as a global variable, typically \( es.tax_{<\text{force name}>} \).

**Infinite Elements**

The Infinite Elements node imposes a coordinate transformation to the selected domain that effectively moves one or more sides of the domain to infinity. Infinite elements are used for the modeling of open boundary problems. A default Charge Conservation node or Ampere’s Law and Current Conservation node is also added.

For the Magnetic and Electric Fields interface, you can also right-click to add additional features. To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button (†) on the Model Builder and then select the applicable option.

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (†) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.
See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**DOMAIN SELECTION**
Select the domains where you want to use infinite elements.

**GEOMETRIC SETTINGS**
Select the type of infinite element scaling to use from the Type list. The options are Cartesian, Cylindrical, Spherical, and General (may be less depending on the spatial dimensions in the model).

In addition, enter the following geometric settings for cylindrical and spherical infinite elements:

- If you select Spherical in 3D or Cylindrical in 2D, enter the components of the Center coordinate \( r_0 \) (3 in 3D, 2 in 2D) in the associated edit fields.
- If you select Cylindrical in 3D, enter the components of the Center coordinate \( r_0 \) and the Center axis direction \( r_{\text{axis}} \) in the associated edit fields.

**PARAMETERS**
To display this section, click the Show button (ckeck) and select Advanced Physics Interface Options. Adjust the two parameters affecting the coordinate transformation—Physical width and Pole distance. Both use default values that should work well for most cases.

The Physical width parameter sets the modeled width of the infinite element region, which typically is a large value. The default value is 1000 times the characteristic distance for the geometry, \( d_{\text{GeomChar}} \). The parameter Pole distance is a tuning parameter that controls the nature of the coordinate transform. The default value is 5 times the average thickness, \( \text{avgDelta} \).

**Manual Scaling**

The Manual Scaling node has no effect unless the type of the infinite element node is General (the software disables the node if you select another type). The settings for manual scaling provide manual control of the stretching in the infinite element domain.

To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button (ckeck) on the Model Builder and then select the applicable option.
SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (show) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

DOMAIN SELECTION
Select the domains where you want to define the infinite element using manual scaling. By default, this node inherits the selection from its parent node, and you can only use a selection that is a subset of the parent node’s selection.

SCALING PARAMETERS
For manual scaling you can control the following parameters:

- The Scaling direction $\mathbf{a}_{rel}$, which is a vector that sets the direction from the interface between the infinite element and the “real geometry” to the outer boundary of the infinite element domain.
- The Geometric width $\Delta_r$ (default value: 1 m), which sets the width of the region.
- The Coordinate at interface $\mathbf{r}_I$, which sets an arbitrary coordinate at the interface.

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.

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (show) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.
button (equation view) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Domain Selection**
Select the domains where you want to define an initial value.

**Initial Values**
Enter a value or expression for the initial value of the Electric potential \( V \). The default value is 0 V.

*Boundary Conditions for the Electrostatics Interface*

**Exterior Boundaries**
The following exterior boundary conditions are available:

- **Ground**—also available for edges and points from the Edges (3D) and Points (2D and 3D) submenus
- **Electric Potential**—also available for edges and points from the Edges (3D) and Points (2D and 3D) submenus
- **Surface Charge Density**
- **Dielectric Shielding**
- **Terminal**
- **Distributed Capacitance**
- **Zero Charge** - the default boundary condition
- **Displacement Field**
- **Periodic Condition**
The relevant interface condition at interfaces between different media is

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

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

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

In addition, the following interior boundary conditions are available:

- Ground
- Electric Potential
- Surface Charge Density
- Zero Charge
- Thin Low Permittivity Gap
- Dielectric Shielding
- Terminal
- Distributed Capacitance

For axisymmetric models, COMSOL Multiphysics takes the axial symmetry boundaries \((r = 0)\) into account and automatically adds an Axial Symmetry feature to the model that is valid on the axial symmetry boundaries only.

Pairs for the Electrostatics Interface

The following are available from the Pairs submenu.

- Ground
- Electric Potential
- Surface Charge Density
- Dielectric Shielding
- Terminal
- Distributed Capacitance
- Zero Charge
- Displacement Field
- Floating Potential
- Continuity

Ground

The Ground node is the default boundary condition and implements ground as the boundary condition \(V = 0\). Ground means that there is a zero potential on the
boundary. This boundary condition is also applicable at symmetry boundaries where the potential is known to be antisymmetric with respect to the boundary.

To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button (\[\text{x}\]) on the Model Builder and then select the applicable option.

**Show or Hide Options for Physics Feature Nodes**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\[\text{x}\]) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Boundary, Edge, or Point Selection**

Select the geometric entity (boundaries, edges, or points) where you want to apply a ground (zero potential) boundary condition. For some interfaces, also select additional Ground features from the Edges (3D models) or Points (2D and 3D models) submenus.

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

**Pair Selection**

If Ground is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

**Constraint Settings**

To display this section, click the Show button (\[\text{x}\]) and select Advanced Physics Interface Options. See Show Advanced Physics Interface Options in the COMSOL Multiphysics User’s Guide.

Select a Constraint type—Bidirectional, symmetric or Unidirectional. If required, select the Use weak constraints check box.
Electric Potential

The Electric Potential node provides an electric potential $V_0$ as the boundary condition $V = V_0$. Because you are solving for the electric potential in this interface, you typically define the value of the potential at some part of the geometry. For some interfaces, also select additional Electric Potential features from the Edges (3D models) or Points (2D and 3D models) submenus.

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (¶) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

BOUNDARY, EDGE, OR POINT SELECTION

Select the geometric entity (boundaries, edges, or points) where you want to apply an electric potential as the boundary condition.

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

PAIR SELECTION

If Electric Potential is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

ELECTRIC POTENTIAL

Enter the value or expression for the Electric potential $V_0$ (SI unit: V).
Surface Charge Density

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

\[-\mathbf{n} \cdot \mathbf{D} = \rho_s, \quad \mathbf{n} \cdot (\mathbf{D}_1 - \mathbf{D}_2) = \rho_s\]

You specify the surface charge density \(\rho_s\) at an outer boundary or at an interior boundary between two nonconducting media.

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

Show or Hide Options for Physics Feature Nodes

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button \(\text{Show} \) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

Boundary Selection

Select the boundaries where you want to apply a surface charge density.

Pair Selection

If Surface Charge Density is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

Surface Charge Density

Enter the value or expression for the Surface charge density \(\rho_s\) (SI unit: C/m²).

To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button \(\text{Show} \) on the Model Builder and then select the applicable option.
**Dielectric Shielding**

The **Dielectric Shielding** node provides a dielectric shielding boundary condition. It describes a thin layer with thickness $d_s$ and a bulk relative permittivity $\varepsilon_r$ that shields the electric field:

$$\mathbf{n} \cdot \mathbf{D} = -\nabla_t \cdot \varepsilon_0 \varepsilon_r d_s \nabla_t V$$

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

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (¶) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and Expanding Advanced Feature Nodes and Sections** for more information.

**BOUNDARY OR EDGE SELECTION**

Select the boundaries or edges (3D models) where you want to apply a dielectric shielding as the condition.

**PAIR SELECTION**

If **Dielectric Shielding** is selected from the **Pairs** submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

**COORDINATE SYSTEM SELECTION**

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

The default is to take the Relative permittivity $\varepsilon_r$ (unitless) values from material. If User defined is selected, select Isotropic, Diagonal, Symmetric, or Anisotropic and enter values or expressions in the field or matrix.

**THIN LAYER**

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

**Terminal**

The **Terminal** node provides a boundary condition for connection to external circuits or with a specified voltage or charge. By specifying zero charge, a floating potential condition is obtained. See also Lumped Parameters for more information.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (¶) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**

Select the boundaries that you want to model as terminals connected to external circuits or an external charge or voltage.

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

**PAIR SELECTION**

If **Terminal** is selected from the **Pairs** submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.
**Terminal**

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

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

- **Voltage** to enter an electric potential \( V_0 \) (SI unit: V).
- **Charge** to enter a charge \( Q_0 \) (SI unit: C). The default is zero charge for an electrode at floating potential.
- **Circuit** to specify a terminal connected to an external circuit.

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

**Floating Potential**

The **Floating Potential** node is used when modeling a metallic electrode at floating potential. For circuit connections use the **Terminal** feature instead.

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

**Show or Hide Options for Physics Feature Nodes**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (🔧) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and Expanding Advanced Feature Nodes and Sections** for more information.

**Boundary Selection**

Select the boundaries where you want to define the floating electrode.

For the **Electric Currents, Shell** interface, you select edges (3D) or points (2D) instead of boundaries.
PAIR SELECTION
If Floating Potential is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

FLOATING POTENTIAL
Specify an optionally non zero Terminal charge $Q_0$ (SI unit: C).

For the Magnetic and Electric Fields and Electric Currents, Shell interfaces, enter a Terminal current $I_0$ (SI unit: A). Specify zero current for a disconnected electrode.

**Displacement Field**

The Displacement Field node provides 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.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ($\pi$) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**

Select the boundaries where you want to use the normal component of the displacement field as the boundary condition.

**PAIR SELECTION**

If Displacement Field is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.
**COORDINATE SYSTEM SELECTION**

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

**DISPLACEMENT FIELD**

Enter the coordinates of the Boundary displacement field $D_0$ (SI unit: C/m$^2$).

**Distributed Capacitance**

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

$$-n \cdot D = \varepsilon_0 \varepsilon_r L \frac{V_{\text{ref}} - V}{d_L} \quad n \cdot (D_1 - D_2) = \varepsilon_0 \varepsilon_r L \frac{V_{\text{ref}} - V}{d_L}$$

You can use this boundary condition to model a thin sheet or film of a dielectric material. The sheet has the relative permittivity $\varepsilon_r L$ and the surface thickness $d_L$, and it is connected to the reference potential $V_{\text{ref}}$.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (¶) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**

Select the boundaries where you want to apply a distributed capacitance.

**PAIR SELECTION**

If Distributed Capacitance is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.
DISTRIBUTED CAPACITANCE

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

Periodic Condition

The Periodic Condition node defines periodicity or antiperiodicity between two boundaries. You can also 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 node. With this node you can manually specify which boundaries constitute the source and destination surfaces. To add the node, right-click the Periodic Condition node and select Destination Selection.

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

BOUNDARY SELECTION

Select the boundaries where you want to apply a periodic condition.

PERIODIC CONDITION

Select a Type of periodicity—Continuity or Antiperiodicity.

Select a Constraint type—Bidirectional, symmetric or Unidirectional. If required, select the Use weak constraints check box.
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.

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

Show or Hide Options for Physics Feature Nodes

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\( \text{Show} \)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

Boundary Selection

Select the boundaries where you want to apply a zero charge condition.

Pair Selection

If Zero Charge is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

Thin Low Permittivity Gap

Use the Thin Low Permittivity Gap condition

\[
\mathbf{n} \cdot \mathbf{D}_1 = \frac{\varepsilon_0 \varepsilon_r L}{d_L} (V_1 - V_2)
\]

\[
\mathbf{n} \cdot \mathbf{D}_2 = \frac{\varepsilon_0 \varepsilon_r L}{d_L} (V_2 - V_1)
\]
to model a thin gap of a material with a small permittivity compared to the adjacent
domains. The layer has the thickness $d_L$ and the relative permittivity $\varepsilon_rL$. The indices
1 and 2 refer to the two sides of the boundary.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution
sections are displayed on a feature node Settings window by default. You can also click
the Expand Sections button on the Model Builder to always show some sections in an
expanded view, or go to these menus to hide options as required. Click the Show
button ( ) on the Model Builder and then select Equation View to display the Equation
View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and
Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**

Select the boundaries where you want to apply a thin low permittivity gap condition.

**THIN LOW PERMITTIVITY GAP**

Enter a Thickness $d$ (SI unit: m).

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

**Continuity**

The Continuity node provides continuity in the field variables across a boundary
between parts in an assembly where you have created a pair. See also Identity and
Contact Pairs in the COMSOL Multiphysics User’s Guide.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution
sections are displayed on a feature node Settings window by default. You can also click
the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( ) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Boundary Selection**
Select individual boundaries in an existing identity pair. This pair first has to be created.

**Pair Selection**
When Continuity is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

**Line Charge**
In 3D specify line charges along the edges of a geometry. To add this feature, right-click the Electrostatics node and select Edges>Line Charge.

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

**Show or Hide Options for Physics Feature Nodes**
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( ) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Edge Selection**
Select the edges where you want to add a line charge.

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

It is possible to add point charges to both 2D and 3D models. To add this feature, right-click the **Electrostatics** node and select **Points>Point Charge**.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (●) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and Expanding Advanced Feature Nodes and Sections** for more information.

**POINT SELECTION**

Select the points where you want to add a point charge.

**POINT CURRENT SOURCE**

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

---

**Electrostatic Point Dipole**

It is possible to add point dipoles to both 2D and 3D models. To add this feature, right-click the **Electrostatics** node and select **Points>Electrostatic Point Dipole**. This 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.

To display additional features for the physics interface feature nodes (and the physics interfaces), click the **Show** button (●) on the **Model Builder** and then select the applicable option.
SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( Show button) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

POINT SELECTION

Select the points where you want to add an electrostatic point dipole.

DIPOLE SPECIFICATION

Select a Dipole specification—Magnitude and direction or Dipole moment.

DIPOLE PARAMETERS

• If Magnitude and direction is selected under Dipole Specification, enter coordinates for the Electric dipole moment direction \( \vec{n}_p \) and the Electric dipole moment, magnitude \( p \) (SI unit: Cm).

• If Dipole moment is selected under Dipole Specification, enter coordinates for the Electric dipole moment \( p \) (SI unit: Cm).
The Electric Currents Interface

The Electric Currents interface ( ), found under the AC/DC branch ( ) in the Model Wizard, has the equations, boundary conditions, and current sources for modeling steady electric currents in conductive media, solving for the electric potential. Current Conservation is the main feature, which adds the equation for the electric potential and provides a settings window for defining the electrical conductivity as well as the constitutive relation and its associated material properties such as the relative permittivity.

For a more extensive introduction to the physics and equations implemented by this interface, see the Theory for the Electric Currents Interface.

When you add this interface, these default nodes are also added to the Model Builder—Current Conservation, Electric Insulation (the default boundary condition), and Initial Values. Right-click the Electric Currents node to add other features that implement, for example, boundary conditions and current sources.

To display additional features for the physics interfaces and feature nodes, click the Show button ( ) in the Model Builder and select the applicable section.

**Show More Options for Physics Interfaces and Feature Nodes**

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

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

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

Availability of each feature, and whether it is described for a particular interface or node, is based on the individual physics interface and feature node. For example, the Discretization, Advanced Settings, Consistent Stabilization, and Inconsistent Stabilization sections are often described individually throughout the documentation as there are
unique settings. See Showing and Expanding Advanced Feature Nodes and Sections in the COMSOL Multiphysics User’s Guide for additional links to the relevant documentation.

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

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

DOMAIN SELECTION
Select the domains where you want to define the electric potential and the equations that describe the potential field for conductive media. The default setting is to include all domains in the model.

PHYSICAL MODEL
Select the Porous media and mixtures check box to enable the modeling of electric currents in porous media saturated with a conducting fluid, or a solid matrix with inclusions of another material with different electric properties. Selecting this check box enables features on the Current Conservation page and adds the option to use the Archie’s Law feature.

OUT OF PLANE THICKNESS (2D MODELS ONLY)
Define the out-of-plane thickness $d$ (see Equation 4-1) by entering a value or expression (SI unit: m) in the Thickness edit field. The default value of 1 m is typically not representative for a thin grounding plate, for example. Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D models.

PORT SWEEP SETTINGS
Select the Activate port sweep check box to switch on the port sweep and invoke a parametric sweep over the ports/terminals. Enter a Port parameter name to assign a specific name to the variable that controls the port or terminal number solved for during the sweep. The generated lumped parameters are in the form of capacitance matrix elements. The port/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. See Lumped Parameters for more information.

Select an Output format for the Touchstone export—Magnitude angle, Magnitude (dB) angle, or Real imaginary. Enter a Reference impedance $Z_{ref}$ (SI unit: $\Omega$). The default is 50 $\Omega$.

**DEPENDENT VARIABLES**

The dependent variable (field variable) is for the Electric potential $V$. You can change the name in the corresponding edit field, but the names of fields and dependent variables must be unique within a model.

**DISCRETIZATION**

To display this section, click the Show button («) and select Discretization. Select an Electric potential—Linear, Quadratic (the default), Cubic, Quartic, or (in 2D only) Quintic.

**SEE ALSO**

- Current Conservation
- Archie’s Law
- External Current Density
- Current Source
- Force Calculation and Infinite Elements
- Initial Values
- Boundary Conditions for the Electric Currents Interface
- Pairs for the Electric Currents Interface
- Line Current Source
- Electric Point Dipole
- Point Current Source

**Current Conservation**

The Current Conservation node adds the appropriate current conservation law and has the following sections for defining the related material properties.

To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button («) on the Model Builder and then select the applicable option.
**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button ( ) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and Expanding Advanced Feature Nodes and Sections** for more information.

**DOMAIN SELECTION**

Select the domains where you want to define the electric potential and the continuity equation that describes the potential field.

For the **Electric Currents, Shell** interface, select boundaries instead of domains.

**MODEL INPUTS**

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty. If you add a linear temperature relation for the conductivity, you can then define the source for the temperature \( T \). From the **Temperature** list, select an existing temperature variable (from another physics interface) if available, or select **User defined** to define a value or expression for the temperature (SI unit: K) in the edit field that appears underneath the list.

**COORDINATE SYSTEM SELECTION**

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

**MATERIALS**

This section is available when the **Porous media and mixtures** check box is selected on the **Electric Currents Settings** window (see **Physical Model**).

Select **Material 1** from the list and enter a **Volume fraction** \( \theta_1 \). Select **Material 2** from the list, and its volume fraction is automatically set to \( \theta_2 = 1 – \theta_1 \). The default is to use **Domain material** for both Material 1 and 2.
**CONDUCTION CURRENT**

By default, the Electrical conductivity $\sigma$ (SI unit: S/m) for the media is defined From material.

If User defined is selected, select Isotropic, Diagonal, Symmetric, or Anisotropic depending on the characteristics of the electrical conductivity, and then enter values or expressions in the field or matrix. If you want to use another type of temperature dependence than a linear temperature relation (see below), you can enter any expression for the conductivity as a function of temperature.

Select Linear temperature relation for a temperature-dependent conductivity (which occurs in, for example, Joule heating, which is also called resistive heating). The following equation then describes the conductivity:

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

where $\rho_0$ is the resistivity at the reference temperature $T_0$, $\alpha$ is the temperature coefficient of resistance, which describes how the resistivity varies with temperature.

The default Reference temperature $T_{\text{ref}}$ (SI unit: K), Resistivity temperature coefficient $\alpha$ (SI unit: 1/K), and Reference resistivity $\rho_0$ (SI unit: $\Omega$m) are taken From material, which means that the values are taken from the boundary material.

To specify other values for any of these properties, select User defined from the corresponding list and then enter a value or expression. $T$ is the current temperature, which can be a value that you specify as a model input or the temperature from a heat transfer interface. The definition of the temperature field appears in the Model Inputs section.

*Effective Conductivity*

When the Porous media and mixtures check box is selected on the Electric Currents Settings window (see Physical Model) and Material 1 and Material 2 are defined (see Materials), this section enables you to define the electric conductivities for the two materials and the effective conductivity for the mixture. See also Effective Conductivity in Porous Media and Mixtures for more information.

The default Electrical conductivity for Material 1 and Material 2 uses values From material and is defined based on settings made in the Materials section.

If User defined is selected, enter another value or expression for Material 1 (or Material 2) conductivity $\sigma_1$ (or $\sigma_2$). Select Isotropic to define a scalar value or Diagonal, Symmetric, or Anisotropic to define a tensor value.
Then select an **Effective conductivity** averaging technique—**Volume average**, **conductivity**, **Volume average**, **resistivity**, or **Power law**.

**ELECTRIC FIELD**

See the settings for Electric Field under Charge Conservation for the **Electrostatics** interface.

**Effective Relative Permittivity**

When the **Porous media and mixtures** check box is selected on the **Electric Currents Settings** window (see **Physical Model**) and **Material 1** and **Material 2** are defined (see **Materials**), this section enables you to define the relative permittivity for the two materials and the effective relative permittivity for the mixture. See also **Effective Relative Permittivity in Porous Media and Mixtures** for more information.

The default **Relative permittivity** for **Material 1** and **Material 2** uses values **From material** and is defined based on settings made in the **Materials** section.

If **User defined** is selected, enter another value or expression for **Material 1** (or **Material 2**) relative permittivity $\varepsilon_1$ (or $\varepsilon_2$). Select **Isotropic** to define a scalar value or **Diagonal**, **Symmetric**, or **Anisotropic** to define a tensor value.

Then select an **Effective relative permittivity** averaging technique—**Volume average**, **permittivity**, **Volume average**, **reciprocal permittivity**, or **Power law**.

**Archie’s Law**

The **Archie’s Law** feature adds a current conservation node specially tailored for the conduction of electric currents in saturated (or variably saturated) porous media. It has the following sections for defining the related material properties. See **Archie’s Law Theory** for more information.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show**
button ($\text{Model Builder}$) and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Domain Selection**

Select the domains where you want to define Archie’s law.

**Model Inputs**

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

**Coordinate System Selection**

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

**Conduction Currents**

By default, the Electrical conductivity $\sigma_L$ (SI unit: S/m) for the fluid is defined from material.

If User defined is selected, enter a value or expression. If you want to use another type of temperature dependence than a linear temperature relation, enter any expression for the conductivity as a function of temperature.

Enter a Porosity $\varepsilon_p$ to set up the volume fraction of the fluid. Enter other Archie’s law parameters as required: Cementation exponent ($m$), Saturation exponent ($n$), and Fluid saturation ($S_L$). All are unitless numbers and the defaults are 0.

**Electric Field**

You set up the permittivity of the saturated porous media. See the settings for Electric Field under Charge Conservation for the Electrostatics interface.

**External Current Density**

The External Current Density node adds an externally generated current density $J_e$ (SI unit: A/m$^2$), which appears in Ohm’s law

$$J = \sigma E + J_e$$

and in the equation that the Electric Currents interface defines.
To display additional features for the physics interface feature nodes (and the physics interfaces), click the **Show** button (▼) on the **Model Builder** and then select the applicable option.

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (▼) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**DOMAIN SELECTION**

Select the domains where you want to define an external current density.

For the Electric Currents, Shell interface, select boundaries instead of domains.

**COORDINATE SYSTEM SELECTION**

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

**EXTERNAL CURRENT DENSITY**

Enter the components (x, y, and z components in 3D, for example) of the **External current density** $J_e$ in the corresponding fields.

**Current Source**

The **Current Source** node adds a distributed current source $Q_j$ (SI unit: A/m$^3$) in the equation that the Electric Currents interface defines. Use this feature with caution as it may violate the current conservation law that is inherent in Maxwell-Ampère’s law.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click
the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (-expand) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**DOMAIN SELECTION**
Select the domains where you want to define a current source.

For the Electric Currents, Shell interface, select boundaries instead of domains.

**CURRENT SOURCE**
Enter a value or expression for the Current source $Q_j$ (SI unit: A/m³).

**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 you need to specify more than one set of initial values, you can add additional Initial Values features.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (expand) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**DOMAIN SELECTION**
Select the domains where you want to define an initial value.
**INITIAL VALUES**

Enter a value or expression for the initial value of the *Electric potential* \( V \) (SI unit: V).
The default value is 0 V.

*Boundary Conditions for the Electric Currents Interface*

The relevant interface condition at interfaces between different media and interior boundaries is continuity; that is,

\[
\mathbf{n}_2 \cdot (\mathbf{J}_1 - \mathbf{J}_2) = 0
\]

which is the natural boundary condition.

*Exterior Boundaries*

The following exterior boundary conditions are available:

- **Ground** as described for the *Electrostatics* interface in the *COMSOL Multiphysics User’s Guide*—also available for *Edges* (3D) and *Points* (2D and 3D)
- **Electric Potential** as described for the *Electrostatics* interface in the *COMSOL Multiphysics User’s Guide*—also available for *Edges* (3D) and *Points* (2D and 3D)
- **Normal Current Density**
- **Distributed Impedance**
- **Electric Shielding**
- **Electric Insulation**—the default exterior boundary condition
- **Periodic Condition**

*Interior Boundaries*

In addition, the following interior boundary conditions are available:

- **Boundary Current Source**
- **Ground** as described for the *Electrostatics* interface in the *COMSOL Multiphysics User’s Guide*—also available for *Edges* (3D) and *Points* (2D and 3D)
- **Electric Potential** as described for the *Electrostatics* interface in the *COMSOL Multiphysics User’s Guide*—also available for *Edges* (3D) and *Points* (2D and 3D)
- **Distributed Impedance**
- **Electric Shielding**
• **Terminal** as described for the **Electrostatics** interface in the *COMSOL Multiphysics User’s Guide*

• **Electric Insulation**

• **Contact Impedance and Pair Contact Impedance**

• **Floating Potential** as described for the **Electrostatics** interface in the *COMSOL Multiphysics User’s Guide*

For axisymmetric models, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an **Axial Symmetry** node to the model that is valid on the axial symmetry boundaries only.

**Pairs for the Electric Currents Interface**

The following are available from the **Pairs** submenu.

• **Sector Symmetry**

• **Boundary Current Source**

• **Ground** as described for the **Electrostatics** interface in the *COMSOL Multiphysics User’s Guide*—also available for **Edges** (3D) and **Points** (2D and 3D)

• **Electric Potential** as described for the **Electrostatics** interface in the *COMSOL Multiphysics User’s Guide*—also available for **Edges** (3D) and **Points** (2D and 3D)

• **Electric Shielding**

• **Terminal** as described for the **Electrostatics** interface in the *COMSOL Multiphysics User’s Guide*

• **Electric Insulation**

• **Floating Potential** as described for the **Electrostatics** interface in the *COMSOL Multiphysics User’s Guide*

• **Continuity**

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

**Show or Hide Options for Physics Feature Nodes**
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\textsuperscript{Show}) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Boundary Selection**
Select the boundaries where you want to apply a current source.

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

**Pair Selection**
If Boundary Current Source is selected from the Pairs menu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

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

**Boundary Current Source**
Enter a value or expression for the Surface current source $Q_j$ (SI unit: A/m$^2$).

**Normal Current Density**
The Normal Current Density node is applicable to exterior boundaries that represent either a source or a sink of current. It provides a condition for specifying the normal current density of an inward or outward current flow:

$$-n \cdot J = J_n$$

You then specify the normal current density using the inward current density $J_n$. 
Alternatively, you can use the current density $J_0$ to define the normal current density:

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

The normal current density is positive when the current flows inward toward the edge.

To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button (_changed from $\mathbf{D}$ to $\mathbf{C}$_) on the Model Builder and then select the applicable option.

**Show or Hide Options for Physics Feature Nodes**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (changed from $\mathbf{D}$ to $\mathbf{C}$) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Boundary Selection**

Select the boundaries where you want to apply a current flow as the boundary condition using the normal current density.

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

**Coordinate System Selection**

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

**Normal Current Density**

Select an option from the Type list—Inward current density or Current density.

- If Inward current density is selected, enter a value or expression for the normal current density $J_0$ (SI unit: $A/m^2$). Use a positive value for an inward current flow or a negative value for an outward current flow.
- If Current density is selected, enter values or expressions for the components of the current density in the $J_0$ edit fields.
Distributed Impedance

**Note:** This feature was previously called Distributed Resistance.

The **Distributed Impedance** node adds a distributed impedance boundary condition according to the following equations layer between exterior boundaries (setting \( \mathbf{J}_2 = 0 \)) and interior boundaries. You can use this boundary condition to model a thin sheet of a resistive material, connected to a reference potential \( V_{\text{ref}} \). The layer impedance can be specified either with the bulk material conductivity \( \sigma_s \), the relative permittivity \( \varepsilon_r \) and the layer thickness \( d_s \), or directly with the surface resistance \( \rho_s \) and capacitance \( C_s \). Assuming DC currents, the equation is:

\[
\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = \frac{\sigma_s}{d_s} (V - V_{\text{ref}})
\]

\[
\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = \frac{1}{\rho_s} (V - V_{\text{ref}})
\]

For the frequency domain and time dependent study types, this boundary condition is slightly more sophisticated and accounts also for capacitive coupling. The equations are:

\[
\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = \frac{(\sigma + j \omega \varepsilon_0 \varepsilon_r)}{d_s} (V - V_{\text{ref}})
\]

\[
\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = \left( \frac{1}{r_s} + j \omega C_s \right) (V - V_{\text{ref}})
\]

\[
\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = \frac{1}{d_L} \left( \sigma (V - V_{\text{ref}}) + \varepsilon_0 \varepsilon_r \frac{\partial}{\partial t} (V - V_{\text{ref}}) \right)
\]

\[
\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = \left( \frac{1}{\rho_s} (V - V_{\text{ref}}) + C_s \frac{\partial}{\partial t} (V - V_{\text{ref}}) \right)
\]

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click...
the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (Í) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and Expanding Advanced Feature Nodes and Sections** for more information.

### **Boundary Selection**
Select the boundaries where you want to apply a distributed impedance.

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

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

### **Distributed Impedance**
Enter the **Reference potential** $V_{\text{ref}}$ (SI unit: V).

Select a potentially complex valued **Layer specification** from the list—**Surface impedance** or **Thin layer**.

- If **Surface impedance** is selected, enter values or expressions for the **Surface resistance** $\rho_s$ (SI unit: $\Omega \cdot m^2$) and for the **Surface capacitance** $C_s$ (SI unit: $F/m^2$).
- If **Thin layer** is selected, enter values or expressions for **Electrical conductivity** $\sigma$ (SI unit: S/m), **Relative permittivity** $\varepsilon_r$ and **Surface thickness** $d_s$ (SI unit: m). The default value for the surface thickness is $5 \cdot 10^{-3}$ m (5 mm).

### **Electric Shielding**
The **Electric Shielding** node provides an electric shielding boundary condition. It describes a thin layer of a highly conductive medium that shields the electric field. The sheet has the electrical conductivity $\sigma_s$ 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

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = -\mathbf{V}_t \cdot d(\sigma_s \mathbf{V}_t V)$$
For the frequency domain and time-dependent study types, also displacement currents are accounted for via the bulk relative permittivity of the sheet; $\varepsilon_{rs}$ and the conservation laws change to:

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = -V_t \cdot d((\sigma_s + j\omega\varepsilon_0\varepsilon_{rs})V_t V)$$

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = -V_t \cdot d\left(\sigma_s V_t V + \varepsilon_0\varepsilon_{rs}\frac{\partial}{\partial t} V_t V\right)$$

For the Electric Currents, Shell interface, the equivalent Wire cross-section area is the shell thickness $d$, multiplied by the layer thickness $d_L$.

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = -V_t \cdot d d_L((\sigma_L + j\omega\varepsilon_0\varepsilon_{rl})V_t V)$$

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = -V_t \cdot d d_L\left(\sigma_L V_t V + \varepsilon_0\varepsilon_{rl}\frac{\partial}{\partial t} V_t V\right)$$

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

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\(\text{Show}\)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**

Select the boundaries where you want to apply an electric shielding as the boundary condition.

For the Electric Currents, Shell interface, you select edges instead of boundaries.
**MODEL INPUT**

Any model inputs (such as temperature for a temperature-dependent electrical conductivity) appear here.

**PAIR SELECTION**

If **Electric Shielding** is selected from the Pairs menu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

**CONDUCTION CURRENT**

The default **Electrical conductivity** of the boundary comes from material as defined on the domain.

If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** from the list then enter a different value or expression in the field or matrix.

Select **Linearized resistivity** to define the electric resistivity (and conductivity) as a linear function of temperature. The following equation then describes the conductivity:

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

where $\rho_0$ is the resistivity at the reference temperature $T_0$, $\alpha$ is the temperature coefficient of resistance, which describes how the resistivity varies with temperature. $T$ is the current temperature, which can be a value that you specify as a model input or the temperature from a heat transfer interface. The definition of the temperature field appears in the **Model Inputs** section.

If **Linearized resistivity** is selected, by default, the **Reference temperature** $T_{ref}$ (SI unit: K), **Resistivity temperature coefficient** $\alpha$ (SI unit: 1/K), and **Reference resistivity** $\rho_0$ (SI unit: $\Omega$m) values are taken from material. Select **User defined** to enter different values or expressions.

**ELECTRIC FIELD**

By default, the **Relative permittivity** $\varepsilon_r$ (unitless) is taken from material. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** from the list then enter a different value or expression in the field or matrix.

**THIN LAYER**

Enter a value or expression for the **Surface thickness** $d_s$ (SI unit: m).

For the **Electric Currents, Shell** interface, enter a value for the **Wire cross-section area** (SI unit: m$^2$).
Electric Insulation

Electric Insulation is the default boundary condition and this feature adds electric insulation as the boundary condition:

\[ \mathbf{n} \cdot \mathbf{J} = 0 \]

This boundary condition means that no electric current flows into the boundary. At interior boundaries, it means that no current can flow through the boundary and that the electric potential is discontinuous across the boundary. It is also applicable at symmetric boundaries where the potential is known to be symmetric with respect to the boundary.

To add electric insulation to an interior boundary, add an Electric Insulation node in addition to the one that represents the default boundary condition. Electric insulation as the default boundary condition is not applicable to interior boundaries.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\( \text{Show} \)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**

Select the boundaries where you want to apply electric insulation.

For some interfaces, All boundaries are selected by default and can not be changed. For the Electric Currents, Shell interface, you select edges (3D) or points (2D) instead of boundaries.

**PAIR SELECTION**

If Electric Insulation is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.
Periodic Condition

See Periodic Condition as described for the Electrostatics interface in the COMSOL Multiphysics User’s Guide.

Contact Impedance and Pair Contact Impedance

**Note:** This feature was previously called Contact Resistance.

Use the Contact Impedance boundary condition on interior boundaries to model a thin layer of resistive material. You can also add it as a pair.

\[
\mathbf{n} \cdot \mathbf{J}_1 = \frac{\sigma_s}{d_s} (V_1 - V_2)
\]

\[
\mathbf{n} \cdot \mathbf{J}_2 = \frac{\sigma}{d_s} (V_2 - V_1)
\]

\[
\mathbf{n} \cdot \mathbf{J}_1 = \frac{1}{\rho_s} (V_1 - V_2)
\]

\[
\mathbf{n} \cdot \mathbf{J}_2 = \frac{1}{\rho_s} (V_2 - V_1)
\]

The layer impedance can be specified either with the bulk material conductivity \(\sigma_s\), the relative permittivity \(\varepsilon_r\) and the layer thickness \(d_s\), or directly with the surface resistance \(\rho_s\) and capacitance \(C_s\). The indices 1 and 2 refer to the two sides of the boundary.

For the frequency domain and time-dependent study types, this boundary condition is slightly more sophisticated and accounts also for capacitive coupling. The corresponding equations are given below:
To display additional features for the physics interface feature nodes (and the physics interfaces), click the \textit{Show} button \( \text{(1) Show} \) on the \textit{Model Builder} and then select the applicable option.

\textbf{SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES}

For most physics interface feature nodes, the \textit{Equation} and \textit{Override and Contribution} sections are displayed on a feature node \textit{Settings} window by default. You can also click the \textit{Expand Sections} button on the \textit{Model Builder} to always show some sections in an expanded view, or go to these menus to hide options as required. Click the \textit{Show} button \( \text{(1) Show} \) on the \textit{Model Builder} and then select \textit{Equation View} to display the \textit{Equation View} node under all physics interface nodes in the \textit{Model Builder}.

See the description for each physics interface for more links or go to \textit{Showing and Expanding Advanced Feature Nodes and Sections} for more information.

\textbf{BOUNDARY SELECTION}

Select the boundaries where you want to apply a contact resistance.

\textbf{PAIR SELECTION}

If \textit{Pair Contact Impedance} is selected from the \textit{Pairs} submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.
CONTACT IMPEDANCE
Select a potentially complex valued Layer specification from the list—Surface impedance or Thin layer.

- If Surface impedance is selected, enter values or expressions for the Surface resistance \( \rho_s \) (SI unit: \( \Omega \text{m}^2 \)) and for the Surface capacitance \( C_s \) (SI unit: \( \text{F/m}^2 \)).
- If Thin layer is selected, enter values or expressions for Electrical conductivity \( \sigma \) (SI unit: \( \text{S/m} \)), Relative permittivity \( \varepsilon_r \) and Surface thickness \( d_s \) (SI unit: m). The default value for the surface thickness is \( 5 \times 10^{-3} \text{ m (5 mm)} \).

Sector Symmetry
Select Sector Symmetry at interfaces between rotating objects where sector symmetry is used. It is only available for pairs. See also Identity and Contact Pairs in the COMSOL Multiphysics User’s Guide.

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\( \text{Show} \)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

BOUNDARY SELECTION
Select individual boundaries in an existing identity pair. This pair first has to be created.

PAIR SELECTION
When Sector Symmetry is selected from the Pairs menu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

SECTOR SETTINGS
Enter the Number of sectors (must be <50).
Select a Type of periodicity—Continuity or Antiperiodicity.

**Continuity**

The Continuity feature provides continuity in the field variables across a boundary between parts in an assembly where you have created a pair.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( çünkü ) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**

Select individual boundaries in an existing identity pair. This pair first has to be created.

**PAIR SELECTION**

When Continuity is selected from the Pairs menu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

**Line Current Source**

In 3D you can specify line sources along the edges of a geometry. To add this feature, right-click the Electric Currents node and select Edges>Line Current Source.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click
the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (↵) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and Expanding Advanced Feature Nodes and Sections** for more information.

**EDGE SELECTION**
Select the edges where you want to add a current source.

**LINE CURRENT SOURCE**
Enter a value or expression to apply a **Line current source** \( Q_j \) (SI unit: \( \text{A/m} \)) to edges. This source represents electric current per unit length.

**Point Current Source**
It is possible to add point sources to both 2D and 3D models. To add this feature, right-click the **Electric Currents** node and select **Points>Point Current Source**.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**
For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (↵) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and Expanding Advanced Feature Nodes and Sections** for more information.

**POINT SELECTION**
Select the points where you want to add a current source.

**POINT CURRENT SOURCE**
Enter a value or expression to apply a **Point current source** \( Q_j \) (SI unit: \( \text{A} \)) to points. This source represents an electric current flowing out of the point.
Electric Point Dipole

The Electric Point Dipole node is available for 2D and 3D models. This 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.

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

Show or hide options for physics feature nodes

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (Visible) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

Point selection

Select the points where you want to add an electrostatic point dipole.

Dipole specification

Select a Dipole specification—Magnitude and direction or Dipole moment.

Dipole parameters

- If Magnitude and direction is selected under Dipole Specification, enter coordinates for the Electric current dipole moment direction $n_p$ and the Electric current dipole moment, magnitude $p$ (SI unit: A·m).
- If Dipole moment is selected under Dipole Specification, enter the components of the Electric current dipole moment $p$ (SI unit: A·m).
The Electric Currents, Shell Interface

The Electric Currents, Shell interface provides the equations, boundary conditions, and current sources for modeling steady electric currents in thin current-conducting shells, solving for the electric potential. Current Conservation is the main feature, which adds the equation for the electric potential and provides a settings window for defining the electrical conductivity as well as the constitutive relation and its associated material properties such as the relative permittivity.

For a more extensive introduction to the physics and equations implemented by this interface, see the Theory for the Electric Currents, Shell Interface.

When you add this interface, these default nodes are also added to the Model Builder—Current Conservation, Electric Insulation (the default edge or point condition), and Initial Values. Right-click the Electric Currents node to add other features that implement, for example, edge or point conditions and current sources.

Note: Except where described below, the majority of the Settings windows are the same as for the Electrostatics and Electric Currents interfaces as referenced.

To display additional features for the physics interfaces and feature nodes, click the Show button in the Model Builder and select the applicable section.

SHOW MORE OPTIONS FOR PHYSICS INTERFACES AND FEATURE NODES

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

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

For most physics feature nodes, both the Equation and Override and Contribution sections are always available. Click the Show button and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.
Availability of each feature, and whether it is described for a particular interface or node, is based on the individual physics interface and feature node. For example, the Discretization, Advanced Settings, Consistent Stabilization, and Inconsistent Stabilization sections are often described individually throughout the documentation as there are unique settings. See Showing and Expanding Advanced Feature Nodes and Sections in the COMSOL Multiphysics User’s Guide for additional links to the relevant documentation.

**INTERFACE IDENTIFIER**

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

The default identifier (for the first interface in the model) is *ecs*.

**BOUNDARY SELECTION**

Select the boundaries (shells) where you want to define the electric potential and the equations that describe the potential field for conductive media. The default setting is to include all boundaries in the model.

**OUT-OF-PLANE THICKNESS (2D ONLY)**

Enter a value or expression for the Thickness $d$ (SI unit: m). The default value is 1 m.

**SURFACE THICKNESS**

Define the surface thickness $d_s$ by entering a value or expression (SI unit: m) in the Surface Thickness edit field. The default value is 1 cm.

**PORT SWEEP SETTINGS**

When activated this invokes a parametric sweep over the ports/terminals in addition to the automatically generated frequency sweep. Tick the Activate port sweep check box to switch on the port sweep. The generated lumped parameters is in the form of an impedance or admittance matrix depending on the port/terminal settings which consistently must be of either fixed voltage or fixed current type. The Port parameter name input field assigns a specific name to the variable that controls the port number solved for during the sweep. The lumped parameters are subject to Touchstone file export. File name and path are entered in an input field.
**DEPENDENT VARIABLES**

The dependent variable (field variable) is for the Electric potential $V$. You can change the name in the corresponding edit field, but the names of fields and dependent variables must be unique within a model.

**DISCRETIZATION**

To display this section, click the Show button and select Discretization. Select an element order for the Electric Potential—Linear, Quadratic (the default), Cubic, Quartic, or (in 2D only) Quintic.

**SEE ALSO**

- Initial Values
- Boundary Conditions for the Electric Currents, Shell Interface
- Edge (3D) or Point (2D) Conditions

---

**Initial Values**

Initial Values 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. If you need to specify more than one set of initial values, you can add additional Initial Values features from the Other menu when you right-click the main feature for the physics interface.

**BOUNDARY SELECTION**

Select the boundaries where you want to define an initial value.

**INITIAL VALUES**

Enter a value or expression for the initial value of the electric potential $V$ in the Electric potential edit field. The default value is 0 V.

---

**Boundary Conditions for the Electric Currents, Shell Interface**

The following boundary conditions are described for the Electric Currents interface. The only difference is that you select boundaries instead of domains for each feature—Current Conservation, External Current Density, and Current Source. See The Electric Currents Interface for details.
**Edge (3D) or Point (2D) Conditions**

The relevant interface condition at interfaces between different media and interior edges/points is continuity; that is,

\[ \mathbf{n}_2 \cdot (\mathbf{J}_1 - \mathbf{J}_2) = 0 \]

which is the natural edge/point condition.

**Exterior Edges or Points**

The following edge conditions (point conditions in 2D) are available on exterior edges (points) and correspond to the boundary conditions in the standard Electric Currents or Electrostatics interfaces:

See [The Electrostatics Interface](#) for these features:

- Ground (also available for points in 3D)
- Electric Potential (also available for points in 3D)
- Terminal

See [The Electric Currents Interface](#) for these features:

- Normal Current Density
- Distributed Impedance
- Electric Shielding
- Electric Insulation - the default edge/point condition

**Interior Edges or Points**

In addition, the following boundary conditions are available on interior edges/points. These features are as described for the Electric Currents or Electrostatics interfaces. The difference is that you can select edges (3D) or points (2D) instead of boundaries.

See [The Electrostatics Interface](#) for these features:

- Electric Potential
- Ground
- Floating Potential. One further difference is you specify an optionally non zero current \( I_0 \) in the Terminal Current field.
- Terminal

See [The Electric Currents Interface](#) for these features:

- Boundary Current Source
• Electric Insulation
• Distributed Impedance
• Electric Shielding. One further difference is that you enter information for the wire cross section area (SI unit: m²). The default value is 1 cm².
• Contact Resistance
Theory of Electric Fields

COMSOL Multiphysics includes physics interfaces for the modeling of static electric fields and currents. Physics interfaces for the modeling of dynamic, quasi-static (that is, without including wave propagation effects) electric fields and currents are available in the AC/DC Module and MEMS Module. What physics interface and study type to select for a particular modeling situation requires a basic understanding of the charge dynamics in conductors.

This section is a brief introduction to Charge Relaxation Theory. After reading it, you should be more confident when deciding what physics interface and study type to use, depending on the material parameters and characteristic time scales involved.

Charge Relaxation Theory

The different physics interfaces involving only the scalar electric potential can be interpreted in terms of the charge relaxation process. The fundamental equations involved are Ohm’s law (\( \mathbf{J} = \sigma \mathbf{E} \)) the equation of continuity

\[
\frac{\partial \mathbf{J}}{\partial t} + \nabla \cdot \mathbf{J} = 0
\]

and Gauss’ law

\[
\nabla \cdot (\varepsilon \mathbf{E}) = \rho
\]

By combining these, one can deduce the following differential equation for the space charge density in a homogeneous medium

\[
\frac{\partial \rho}{\partial t} + \frac{\sigma}{\varepsilon} \rho = 0
\]

This equation has the solution

\[
\rho(t) = \rho_0 e^{-t/\tau}
\]

where

\[
\tau = \frac{\varepsilon}{\sigma}
\]
is called the charge relaxation time. For a good conductor like copper, \( \tau \) is of the order of \( 10^{-19} \) s whereas for a good insulator like silica glass, it is of the order of \( 10^3 \) s. For a pure insulator, it becomes infinite.

When modeling real world devices, there is not only the intrinsic time scale of charge relaxation time but also an external time scale \( t \) at which a device is energized or the observation time. It is the relation between the external time scale and the charge relaxation time that determines what physics interface and study type to use. The results are summarized in Table 4-1 below.

<table>
<thead>
<tr>
<th>CASE</th>
<th>PHYSICS INTERFACE</th>
<th>STUDY TYPE</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \tau \gg t )</td>
<td>Electrostatics</td>
<td>Stationary</td>
</tr>
<tr>
<td>( \tau \ll t )</td>
<td>Electric Currents</td>
<td>Stationary</td>
</tr>
<tr>
<td>( \tau \sim t )</td>
<td>Electric Currents</td>
<td>Time Dependent or Frequency Domain (in AC/DC Module or MEMS Module)</td>
</tr>
</tbody>
</table>

**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 given model input and the best approach is to solve the Electrostatics formulation using the electric potential \( V \).

By combining the definition of the potential with Gauss’ law, you can derive the classical Poisson’s equation. Under static conditions, the electric potential \( V \) is defined by the equivalence \( E = -\nabla V \). Using this together with the constitutive relation \( D = \varepsilon_0 E + P \) between \( D \) and \( E \), you can rewrite Gauss’ law as a variant of Poisson’s equation

\[
\nabla \cdot (\varepsilon_0 \nabla V - P) = \rho
\]

This equation is used in the Electrostatics interface. It is worth noting that Gauss’ law does not require the charge distribution to be static. Thus, provided dynamics are slow enough that induced electric fields can be neglected and hence a scalar electric potential is justified, the formulation can be used also in the Time Dependent study type. That typically involves either prescribing the charge dynamics or coupling a separate formulation for this. Such separate charge transport formulations can be found in the Plasma Module and the Chemical Reaction Engineering Module.
SECOND CASE: $\tau << \tau$  

If the external time scale is long compared to the charge relaxation time, the stationary solution to the equation of continuity has been reached. In a stationary coordinate system, a slightly more general form than above of Ohm’s law states that

$$J = \sigma E + J^e$$

where $J^e$ is an externally generated current density. The static form of the equation of continuity then reads

$$\nabla \cdot J = -\nabla \cdot (\sigma \nabla V - J^e) = 0$$

To handle current sources the equation can be generalized to

$$-\nabla \cdot (\sigma \nabla V - J^e) = Q_j$$

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

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

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

$$-\nabla \cdot ((\sigma + j \omega \epsilon_0) \nabla V - (\mathbf{J}^e + j \omega \mathbf{P})) = 0$$

For the time dependent study type, use the transient equation of continuity

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

and the resulting equation becomes

$$-\nabla \cdot \left(\varepsilon_0 \nabla V + \mathbf{P}\right) - \nabla \cdot (\sigma \nabla V - \mathbf{J}^e) = 0$$
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 may not meet the large skin depth condition. In that case, the electrodes must only be represented as boundary conditions (fixed or floating potential). The interior, metallic domains are not included in the analysis. Obviously, this is only a valid approach for devices where metallic electrodes do not entirely bypass (short circuit) the capacitive/resistive layers. If metallic electrodes short circuit the capacitive/resistive layers, the time evolution of the current is determined by inductive and resistive effects with very little influence from the capacitive layers. Then the Magnetic Fields interface is the appropriate modeling tool.
Theory for the Electrostatics Interface

The Electrostatics interface is available for 3D, 2D in-plane, and 2D axisymmetric models. Applications with Electrostatics Equations include high-voltage apparatus, electronic devices, and capacitors. The term “statics” is not to be interpreted literally—it is the observation time or time scale at which the applied excitation changes is short compared to the charge relaxation time and that the electromagnetic wavelength and skin depth are very large compared to the size of the domain of interest. If you do not know whether to use the Electric Currents or the Electrostatics interface, which both solve for the scalar electric potential $V$, consider using an explicit charge transport model See Charge Relaxation Theory.

Electrostatics Equations

Under static conditions the electric potential, $V$, is defined by the relationship

$$E = -\nabla V$$

Combining this equation with the constitutive relationship $D = \varepsilon_0 E + P$ between the electric displacement $D$ and the electric field $E$, it is possible to represent Gauss’ law as the following equation:

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

In this equation, the physical constant, $\varepsilon_0$ (SI unit: F/m) is the permittivity of vacuum, $P$ (SI unit: C/m$^2$) is the electric polarization vector, and $\rho$ (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, $E$, is tangential to the $xy$-plane. With this symmetry, you solve the same equation as in the 3D case. The interface solves the following equation where $d$ is the thickness in the $z$ direction:

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

The axisymmetric version of the interface considers the situation where the fields and geometry are axially symmetric. In this case the electric potential is constant in the $\phi$ direction, which implies that the electric field is tangential to the $rz$-plane.
Theory for the Electric Currents Interface

The Electric Currents interface solves a current conservation problem for the scalar electric potential $V$ and is available for 3D, 2D in-plane, and 2D axisymmetric models. Electrolysis and the computation of resistances of grounding plates are examples that involve conductive media with electrical conductivity $\sigma$ and electric currents. If you are uncertain whether to use the Electric Currents interface or the Electrostatics interface which both solve for the scalar electric potential $V$, refer to the section on Charge Relaxation Theory.

In this section:
- Electric Currents Equations in Steady State
- Effective Conductivity in Porous Media and Mixtures
- Dynamic Electric Currents Equations
- Effective Relative Permittivity in Porous Media and Mixtures
- Archie’s Law Theory
- Reference for the Electric Currents Interface

Electric Currents Equations in Steady State

When handling stationary electric currents in conductive media you must consider the stationary equation of continuity. In a stationary coordinate system, the point form of Ohm’s law states that

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

where $\sigma$ is the electrical conductivity (SI unit: S/m), and $\mathbf{J}_e$ is an *externally generated current density* (SI unit: A/m$^2$). The static form of the equation of continuity then states

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

To handle *current sources*, you can generalize the equation to

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

where $Q_j$ is the current source density.
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:

\[
-\nabla \cdot (\sigma \nabla V - \mathbf{J}_e) = d \mathbf{Q}_j
\]  

(4-1)

In 2D axisymmetry, the Electric Currents interface considers the situation where the fields and geometry are axially symmetric. In this case the electric potential is constant in the \( \varphi \) direction, which implies that the electric field is tangential to the \( rz \)-plane.

**SEE ALSO**

- Effective Conductivity in Porous Media and Mixtures
- Dynamic Electric Currents Equations
- Effective Relative Permittivity in Porous Media and Mixtures
- Archie’s Law Theory

*Effective Conductivity in Porous Media and Mixtures*

When handling electric currents in porous media or mixtures of solids with different electric properties, you must consider different ways for obtaining the effective conductivity of the mixture.

There are several possible approaches to do this, starting from the values defined by the user, composed by a volume fraction \( \theta_1 \) of material 1, and a volume fraction \( \theta_2 = 1 - \theta_1 \) of material 2.

The effective conductivity \( \sigma \) is then given as input for the electric current conservation specified in Equation 4-1, in the same way of modeling an effective (single phase) material.

**Volume Average, Conductivity**

If the electric conductivities of the two materials are not so different from each other, a simple form of averaging can be used, such as a volume average:

\[
\sigma = \theta_1 \sigma_1 + \theta_2 \sigma_2
\]

here \( \sigma_1 \) is the conductivity of the material 1 and \( \sigma_2 \) is that of material 2. This is equivalent to a “parallel” system of resistivities.
Note that if the conductivities are defined by second order tensors (such as for anisotropic materials), the volume average is applied element by element.

**Volume Average, Resistivity**
A similar expression for the effective conductivity can be used, which mimics a “series” connection of resistivities. Equivalently, the effective conductivity is obtained from

\[
\frac{1}{\sigma} = \frac{\sigma_1}{\sigma_1} + \frac{\sigma_2}{\sigma_2}
\]

Note that if the conductivities are defined by second order tensors, the inverse of the tensors are used.

**Power Law**
A power law gives the following expression for the equivalent conductivity:

\[
\sigma = \sigma_1 \sigma_2
\]

Note that the effective conductivity calculated by Volume Average, Conductivity is the upper bound, the effective conductivity calculated by Volume Average, Resistivity is the lower bound, and the Power Law average is somewhere in between these two.

**See Also**
- Electric Currents Equations in Steady State
- Dynamic Electric Currents Equations
- Effective Relative Permittivity in Porous Media and Mixtures
- Archie’s Law Theory

**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

\[
\nabla \cdot J = \nabla \cdot (\sigma E + J^f) = -j\omega\rho
\]

with the equation

\[
\nabla \cdot D = \rho
\]
and generalized to handle current sources yields the following equation:

\[-\nabla \cdot \left[ \left( \sigma + j\omega \varepsilon_0 \right) \nabla V - \left( \mathbf{J}^e + j\omega \mathbf{P} \right) \right] = Q_j\]

For the transient case, using the transient equation of continuity

\[\nabla \cdot \mathbf{J} = \nabla \cdot \left( \sigma \mathbf{E} + \mathbf{J}^p \right) = -\frac{\partial \rho}{\partial t}\]

and generalized to handle current sources the resulting equation becomes

\[-\nabla \cdot \frac{\partial}{\partial t}(\varepsilon_0 \nabla V + \mathbf{P}) - \nabla \cdot (\sigma \nabla V - \mathbf{J}^e) = Q_j\]  \hspace{1cm} (4-2)

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

\[-\nabla \cdot d\left( \left( \sigma + j\omega \varepsilon_0 \right) \nabla V - \left( \mathbf{J}^e + j\omega \mathbf{P} \right) \right) = dQ_j\]

\[-\nabla \cdot d\frac{\partial}{\partial t}(\varepsilon_0 \nabla V + \mathbf{P}) - \nabla \cdot d(\sigma \nabla V - \mathbf{J}^e) = dQ_j\]

**Effective Relative Permittivity in Porous Media and Mixtures**

When handling electric currents in porous media or mixtures of solids with different electric properties, you must consider different ways for obtaining the effective relative permittivity of the mixture.

There are several possible approaches to do this, starting from the values defined by the user, composed by a volume fraction \(\theta_1\) of material 1, and a volume fraction \(\theta_2 = 1 - \theta_1\) of material 2.

The effective relative permittivity \(\varepsilon_r\) is then given as input for the electric current conservation specified in Equation 4-2, in the same way of modeling an effective (single phase) material.

**Volume Average, Permittivity**

If the relative permittivities of the two materials are not so different from each other, the effective relative permittivity \(\varepsilon_r\) is calculated by simple volume average:

\[\varepsilon_r = \theta_1 \varepsilon_1 + \theta_2 \varepsilon_2\]

here \(\varepsilon_1\) is the relative permittivity of the material 1, and \(\varepsilon_2\) is that of material 2.
Note that if the permittivities are defined by second order tensors (such as for anisotropic materials), the volume average is applied element by element.

**VOLUME AVERAGE, RECIPROCAL PERMITTIVITY**

A similar expression for the effective permittivity can be used, which mimics a “series” connection of resistivities. Equivalently, the effective reciprocal permittivity is obtained from

$$\frac{1}{\varepsilon_r} = \frac{\theta_1}{\varepsilon_1} + \frac{\theta_2}{\varepsilon_2}$$

Note that if the permittivities 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 permittivity:

$$\varepsilon_r = \frac{\varepsilon_1 \varepsilon_2}{\theta_1 \theta_2}$$

Note that 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 in between these two.

**SEE ALSO**

- Electric Currents Equations in Steady State
- Effective Conductivity in Porous Media and Mixtures
- Dynamic Electric Currents Equations
- Archie’s Law Theory

**Archie’s Law Theory**

The electrical conductivity of the materials composing saturated rocks and soils can vary over many orders of magnitude. For instance, in the petroleum reservoirs, normal sea water (or brine) has a typical conductivity of around 3 S/m, whereas hydrocarbons are typically much more resistive and have conductivities in the range 0.1 – 0.01 S/m.

The porous rocks and sediments may have even lower conductivities. In variably saturated soils, the conductivity of air is roughly ten orders of magnitude lower that
the ground water. A simple volume average (of either conductivity or resistivity) in rocks or soils might give different results compared to experimental data.

Since most crustal rocks, sedimentary rocks, and soils are formed by non-conducting materials, Archie (Ref. 1) assumed that electric current are mainly caused by ion fluxes 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 \( \sigma_L \), fluid saturation \( s_L \) and porosity \( \varepsilon_p \):

\[
\sigma = s_L^n \varepsilon_p^m \sigma_L
\]

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 it is close to 2 for sandstones. The lower limit \( m = 1 \) represents a volume average of the conductivities of a fully saturated, insulating (zero conductivity) porous matrix, and a conducting fluid. The saturation coefficient \( n \) is normally close to 2.

**Note:** The ratio \( F = \sigma_L / \sigma \) is called the *formation factor.*

Archie’s Law does not take care of the relative permittivities of either fluids or solids, so the effective relative permittivity of the porous medium is normally consider as \( \varepsilon_r = 1. \)

**SEE ALSO**
- Electric Currents Equations in Steady State
- Effective Conductivity in Porous Media and Mixtures
- Dynamic Electric Currents Equations
- Effective Relative Permittivity in Porous Media and Mixtures
- Archie’s Law Theory

**Reference for the Electric Currents Interface**

Theory for the Electric Currents, Shell Interface

Use the Electric Currents, Shell interface in 3D to model thin shells of conductive media. This physics interface is similar to the 2D Electric Currents interface, solving for the electric potential on 2D surfaces in a 3D geometry. The difference is that the shell does not have to be flat as they obviously are when using the 2D Electric Currents interface. The Electric Currents, Shell interface is also available on boundaries in 2D geometries.

In this section:
- Electric Currents, Shell Equations in Steady State
- Dynamic Electric Currents Equations

Electric Currents, Shell Equations in Steady State

In the static study type, the interface solves the following equation where \( d \) is the thickness (SI unit: m) of the shell:

\[
-\nabla_t \cdot \left[ \sigma \nabla_t V - J^e \right] = dQ_j
\]

\( \sigma \) is the electrical conductivity (SI unit: S/m), \( J^e \) 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 \( \nabla_t \) 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:

\[
-\nabla_t \cdot \left[ \sigma \nabla_t V - (J^e + j\omega P) \right] = dQ_j
\]

For the transient case, the resulting equation becomes

\[
-\nabla_t \cdot \frac{\partial}{\partial t} \left( \varepsilon_0 \nabla_t V + P \right) - \nabla_t \cdot \left[ \sigma \nabla_t V - J^e \right] = dQ_j
\]
This chapter summarizes the functionality of the magnetic field interfaces found under the AC/DC branch ( ) in the Model Wizard.

In this chapter:

- The Magnetic Fields Interface
- The Magnetic Fields, No Currents Interface
- The Rotating Machinery, Magnetic Interface
- Theory of Magnetic and Electric Fields
- Theory for the Magnetic Fields Interface
- Theory for the Magnetic Fields, No Currents Interface
The Magnetic Fields Interface

**Note:** The AC/DC Module enhances the Magnetic Fields interface included with the basic COMSOL Multiphysics license.

The Magnetic Fields interface ( ), found under the AC/DC branch ( ) in the Model Wizard, has the equations, boundary conditions, and external currents for modeling magnetic fields, solving for the magnetic vector potential. The main feature is the Ampère’s Law feature, which adds the equation for the magnetic vector potential and provides an interface for defining the constitutive relation and its associated properties such as the relative permeability. For a more thorough introduction to the equations solved by this physics interface, see the Theory for the Magnetic Fields Interface.

When you add this interface, these default nodes are also added to the Model Builder—Magnetic Fields, Ampère’s Law, Magnetic Insulation (the default boundary condition), and Initial Values. Right-click the Magnetic Fields node to add other features that implement, for example, boundary conditions and external currents.

To display additional features for the physics interfaces and feature nodes, click the Show button ( ) in the Model Builder and select the applicable section.

**SHOW MORE OPTIONS FOR PHYSICS INTERFACES AND FEATURE NODES**

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

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

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

Availability of each feature, and whether it is described for a particular interface or node, is based on the individual physics interface and feature node. For example, the
Discretization, Advanced Settings, Consistent Stabilization, and Inconsistent Stabilization sections are often described individually throughout the documentation as there are unique settings. See Showing and Expanding Advanced Feature Nodes and Sections in the COMSOL Multiphysics User’s Guide for additional links to the relevant documentation.

**INTERFACE IDENTIFIER**

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

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

**DOMAIN SELECTION**

Select the domains where you want to define the magnetic vector potential and the equations that describe the potential field for magnetic fields. The default setting is to include all domains in the model.

**SETTINGS**

The Components list allows you to define the components of the magnetic vector potential that you wish to solve for. This setting is only available in 2D and 2D axially symmetric models. Note that the current vector has the same direction as the magnetic vector potential, so this setting also controls the direction in which applied and induced currents can flow in the model. The default option is to solve for the out-of-plane component only.

In the Solve for list you have the option to solve for Reduced field or Full field. If you choose to solve for the reduced field you can specify a background field expressed as a background magnetic vector potential in the Ab edit field. The total field used in the physics and equations are given by the sum of the reduced and background fields.

**OUT-OF-PLANE THICKNESS (2D MODELS ONLY)**

Define the out-of-plane thickness \( d \) by entering a value or expression (SI unit: m) in the Thickness field. The default value of 1 m is typically not representative for a thin domain. Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D models.
**DISCRETIZATION**

To display this section, click the Show button (on) and select Discretization. Select a Magnetic vector potential—Quadratic (the default), Linear, Cubic, or Quartic.

**SEE ALSO**
- Ampère’s Law
- Gauge Fixing for A-field
- External Current Density
- Velocity (Lorentz Term)
- Force Calculation and Infinite Elements as described for the Electrostatics interface
- Multi-Turn Coil Domain
- Single-Turn Coil Domain
- Coil Group Domain
- Reversed Current Direction
- Initial Values
- Boundary Conditions for the Magnetic Fields Interface
- Pairs for the Magnetic Fields Interface

---

**Ampère’s Law**

The Ampère’s Law feature adds Ampère’s law for the magnetic field and provides an interface for defining the constitutive relation and its associated properties such as the relative permeability as well as electric properties.

Right-click the **Ampère’s Law** node to add a Gauge Fixing for A-Field feature. For some interfaces this feature is added by default.

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

---

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show
button ( ) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Domain Selection**
Select the domains where you want to define the magnetic vector potential and the equation based on Ampère’s law that defines the potential.

**Model Inputs**
This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty. If you add a linear temperature relation for the conductivity, you can then define the source for the temperature $T$. From the Temperature list, select an existing temperature variable (from another physics interface) if available, or select User defined to define a value or expression for the temperature (SI unit: K) in the edit field that appears underneath the list.

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

**Magnetic Field**
Specify the constitutive relation that describe the macroscopic properties of the medium (relating the magnetic flux density $B$ and the magnetic field $H$) and the applicable material properties, such as the relative permeability.

Select one of the following constitutive relations from the Constitutive relation list (the equation for the selected constitutive relation appears under the list):

- Select Relative permeability $\mu_r$ to use the constitutive relation $B = \mu_0 \mu_r H$ (the default). The default uses values From material. If User defined is selected, choose Isotropic, Diagonal, Symmetric, or Anisotropic and enter values or expressions in the field or matrix.
- Select Magnetization to use the constitutive relation $B = \mu_0 H + \mu_0 M$. You then specify $M$, the magnetization vector (SI unit: A/m). Enter its components ($3$ in 3D, $2$ in 2D) in the edit fields in the table that appears under the constitutive equation.
- Select Magnetic losses to describe the relative permeability as a complex-valued quantity: $\mu_r = \mu' + i\mu''$, where $\mu'$ and $\mu''$ are the real and imaginary parts, respectively.
The default uses values From material. If User defined is selected, enter values or expressions for the real and imaginary parts.

This option is not available for the Magnetic Fields, No Currents interface.

- Select HB curve to use a curve that relates magnetic flux density $B$ and the magnetic field $H$ as $|H| = \hat{f}(B)$. The default is to use the value $|H|$, From material. If User defined is selected, enter a value or expression for the magnitude of the magnetic field.

If you are using the Induction Heating interface, this option is not relevant for time harmonic modeling so it should not be selected.

- Select Remanent flux density to use the constitutive relation $B = \mu_0 \mu_r H + B_r$, where $B_r$ is the remanent flux density (the flux density when no magnetic field is present).

In this case you specify $\mu_r$, the relative permeability (unitless), and $B_r$, the remanent flux density (SI unit: T). For the relative permeability, use the $\mu_r$ list: The default uses values From material. If User defined is selected, choose Isotropic, Diagonal, Symmetric, or Anisotropic based on the characteristics of the relative permeability and enter another value or expression in the field or matrix. For the remanent displacement, enter its components (3 in 3D, 2 in 2D) in the $B_r$ table.

**Conduction Current**

By default, the Electrical conductivity $\sigma$ (SI unit: S/m) for the media is defined From material.

If User defined is selected, choose Isotropic, Diagonal, Symmetric, or Anisotropic based on the characteristics of the electrical conductivity, and then enter values or expressions in the field or matrix.

If Linearized resistivity is selected, it defines the electric resistivity (and conductivity) as a linear function of temperature and this equation describes the conductivity:

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

where $\rho_0$ is the resistivity at the reference temperature $T_0$. $\alpha$ 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 $\alpha$ (SI unit: 1/K), and Reference resistivity $\rho_0$ (SI unit: Ω·m) are taken From material, which means that the values are taken from the boundary material.
To specify other values for any of these properties, select **User defined** from the corresponding list and then enter a value or expression. \( T \) is the current temperature, which can be a value that you specify as a model input or the temperature from a heat transfer interface. The definition of the temperature field appears in the **Model Inputs** section.

**ELECTRIC FIELD**

The default **Relative permittivity** \( \varepsilon_r \) (unitless) for the media is **From material** and defined on the shell domain.

If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the permittivity and then enter values or expressions in the field or matrix.

---

**Gauge Fixing for A-field**

Right-click the **Ampère's Law** node to add the **Gauge Fixing for A-Field** subnode. The feature provides gauge fixing by adding an additional potential variable, \( \psi \), 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 \( \mathbf{A} \). For more details on gauge fixing see [Explicit Gauge Fixing/Divergence Constraint](#).

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

---

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (¶) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to [Showing and Expanding Advanced Feature Nodes and Sections](#) for more information.

**DOMAIN SELECTION**

Select the domains where you want to define the gauge-fixing potential \( \psi \).
GAUGE FIXING FOR A-FIELD
Enter a Divergence condition variable scaling $\psi_0$ (SI unit: A/m). The default value is 1 A/m, which means no scaling.

External Current Density

The External Current Density feature adds an externally generated current density $J_e$, which appears on the right-hand side of the equation that the Magnetic Fields interface defines.

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( ) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

DOMAIN SELECTION
Select the domains where you want to define an external current density.

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

EXTERNAL CURRENT DENSITY
Enter a value or expression for each component of the External current density $J_e$ (SI unit: A/m$^2$) in the table.
**Velocity (Lorentz Term)**

The **Velocity (Lorentz term)** feature adds velocity \( v \). The external current is equal to \( \sigma v \times B \). This feature is only valid when:

- solving for both the electric potential and the magnetic vector potential using the Magnetic and Electric Fields interface
- in 2D and 2D axisymmetry when solving for only the out-of-plane component of the magnetic vector potential

**Note:** To use the velocity feature correctly requires deep physical insight. In situations when the moving domain is of bounded extent in the direction of the motion or material properties vary in this direction or it contains magnetic sources that also move, the Lorentz term must not be used. An operational definition of when it can be used is that the moving domain should only contain an induced magnetic source (magnetization + eddy currents) that has to be stationary with respect to the motion. Thus, it cannot be used for modeling projectiles of finite length or projectiles containing magnets. It can be used to model conductive, homogeneous spinning disks (magnetic brakes), magnets over a moving infinite homogenous plane (maglev trains), a flow of homogeneous conducting fluid past a magnet (liquid metal pumps, Hall generators/thrusters). If in doubt, contact support.

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

### Show or Hide Options for Physics Feature Nodes

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (\( \text{Show} \)) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and Expanding Advanced Feature Nodes and Sections** for more information.


**DOMAIN SELECTION**
Select the domains where you want to define the velocity.

**COORDINATE SYSTEM SELECTION**
The global coordinate system is selected by default. The Coordinate system list contains any additional coordinate systems that the model includes.

**VELOCITY (LORENTZ TERM)**
Enter components of the Velocity vector \( \mathbf{v} \) (SI unit: m/s).

**Multi-Turn Coil Domain**
The Multi-Turn Coil Domain feature is available for 2D and 2D axisymmetric models. It adds an externally generated current density to the right-hand side of the equation that the Magnetic Fields interface defines. This feature sets the conductivity of the domain to zero because the induced current in the coil windings is handled in a homogenized manner. The external current density is calculated in different ways, depending on whether you specify a total current or a total voltage.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( ) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**DOMAIN SELECTION**
Select the domains where you want to define the multi-turn coil domain.

**MODEL INPUTS**
This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty.
COORDINATE SYSTEM SELECTION

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

MULTI-TURN COIL DOMAIN

- Enter a **Coil name**.
- The default **Relative permeability** $\mu_r$ (unitless) and **Relative permittivity** $\varepsilon_r$ (unitless) are taken **From material**. If **User defined** is selected for either variable, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the coil and then enter values or expressions in the field or matrix.
- Enter a **Coil conductance** $\sigma_{coil}$ (SI unit: S/m). The default value is approximately the conductance for copper, $6 \cdot 10^7$ S/m.
- Enter the **Number of turns** $N$. the default is 10.
- Enter a **Coil wire cross-section area** $a_{coil}$ (SI unit: m$^2$). The default value is $10^{-6}$ m$^2$.
- Select a **Coil excitation**—**Circuit**, **Voltage**, or **Current**. See below for details.
  - If **Current** is selected, also enter a **Coil current** $I_{coil}$ (SI unit: A). The default value is 1 A.
  - If **Voltage** is selected, also enter a **Coil potential** $V_{coil}$ (SI unit: V). The default value is 1 V.

Current-Driven Coil

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

$$J_e = \frac{NI_{coil}}{A}$$

where $N$ is the number of turns which you have specified and $A$ is the total cross-section area of the coil domain.

Voltage-Driven Coil

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

$$J_e = \frac{N(V_{coil} + V_{ind})}{AR_{coil}}$$
where \( V_{\text{coil}} \) is the applied voltage which you have specified, \( A \) is the total cross-sectional area of the coil domain, \( N \) is the number of turns that you have specified, \( R_{\text{coil}} \) is the total resistance of the coil calculated as

\[
\frac{\int \frac{NL}{\sigma_{\text{coil}}} da_{\text{coil}}}{A}
\]

and \( V_{\text{ind}} \) is the induced voltage calculated by integrating the electric field along the coil.

**Circuit Connection**

By selecting Circuit from the Coil excitation list, you can drive the Multi-Turn Coil Domain feature by a voltage defined in a component in the Electrical Circuit interface.

**Inductance Calculation (2D Axisymmetric Models)**

For 2D axisymmetric models the Multi-Turn Coil Domain feature calculates the inductance for the coil. This value is stored in a global variable \( Lc_{x.mf} \) (available for evaluation in Results), where \( x \) is the name given to the coil in the Coil Name edit field.

**Single-Turn Coil Domain**

The Single-Turn Coil Domain feature is available for 2D and 2D axisymmetric models. It adds an externally generated current density to the right-hand side of the equation that the Magnetic Fields interface defines. This current density is calculated in two different ways, depending on whether the user specifies a total current or a total voltage.

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

**Show or Hide Options for Physics Feature Nodes**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\( \uparrow \)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.
**DOMAIN SELECTION**

Select the domains where you want to define the single-turn coil domain.

**SINGLE-TURN COIL DOMAIN**

Enter a **Coil name**.

Select a **Coil excitation**—**Voltage**, **Current**, or **Circuit**. See below for details.

- If **Voltage** is selected, also enter a **Coil potential** $V_{coil}$ (SI unit: V). The default value is 1 V.
- If **Current** is selected, also enter a **Coil current** $I_{coil}$ (SI unit: A). The default value is 1 A.

**Voltage-Driven Coil**

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

\[ J^e = \frac{\sigma V_{coil}}{L} \]

where $V_{coil}$ is the applied voltage that you have 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.

**Current-Driven Coil**

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

\[ J^c = \frac{\sigma V}{L} \]

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 $I_{coil}$ that you have specified.

**Circuit Connection**

By selecting **Circuit** from the **Coil excitation** list, you can drive the **Single-Turn Coil Domain** feature by a voltage defined in a component in the **Electrical Circuit** physics interface.
**INDUCTANCE CALCULATION (2D AXISYMMETRIC MODELS)**

For 2D axially symmetric models the Single-Turn Coil Domain feature calculates the inductance for the coil. This value is stored in a global variable \( L_{c,x,mf} \) (available for evaluation in **Results**), where \( x \) is the name given to the coil in the **Coil name** edit field.

**Coil Group Domain**

The Coil Group Domain feature is available for 2D and 2D axisymmetric models. It adds an externally generated current density to the right-hand side of the equation that the Magnetic Fields interface defines. This current density is calculated in three different ways, depending on whether you specify a fixed current in each coil turn, a total voltage drop across the coil, or a fixed power into the coil. Right-click to add a Reversed Current Direction node.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (ображення) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and Expanding Advanced Feature Nodes and Sections** for more information.

**DOMAIN SELECTION**

Select the domains where you want to define the coil group domain.

**COIL GROUP DOMAIN**

Enter a **Coil name**.

Select a **Coil excitation**—Voltage, Current, Circuit, or Power.

- If **Voltage** is selected, also enter a **Coil potential** \( V_{coil} \) (SI unit: V). The default is 1 V.
- If **Current** is selected, also enter a **Coil current** \( I_{coil} \) (SI unit: A). The default is 1 A.
- If **Power** is selected, also enter a **Coil power** \( P_{coil} \) (SI unit: W). The default is 1 W.
Voltage-Driven Coil

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

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

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

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

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

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

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

Current-Driven Coil

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

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

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

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

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

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

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

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

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

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

where $N$ is the number of turns in the coil group and $V_{drop}$ is defined as:

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

For more information on the \texttt{realdot} operator, see The Realdot Operator in the COMSOL Multiphysics User’s Guide. The Power option results in a highly nonlinear system of equations that requires special solver settings in order to converge. COMSOL adds these solver settings automatically when you solve a model using this setting.

Circuit Connection

By selecting Circuit from the Coil excitation list, you can drive the Coil Group Domain feature by a current defined in a component in the Electrical Circuit interface.

LUMPED PARAMETER CALCULATIONS (2D AXISYMMETRIC MODELS)

For 2D axially symmetric models the Coil Group Domain feature calculates the impedance, resistance, reactance, inductance, and admittance for the coil. In the following definitions, $V_{drop}$ is defined as the sum of the applied potentials:
Impedance
This value is stored in a global variable \( \text{mf.Z cg}<\text{name}> \) (available for evaluation in Results) where \(<\text{name}>\) is the name given to the coil in the Coil name field. The coil impedance is computed using the formula:

\[
Z = \frac{V_{\text{drop}}}{I_{\text{coil}}}
\]

Resistance
This value is stored in a global variable \( \text{mf.R coil cg}<\text{name}> \) (available for evaluation in Results) where \(<\text{name}>\) is the name given to the coil in the Coil name edit field. The coil resistance is computed using the formula:

\[
R_{\text{coil}} = \text{Re}\left(\frac{V_{\text{drop}}}{I_{\text{coil}}}\right)
\]

Reactance
This value is stored in a global variable \( \text{mf.X cg}<\text{name}> \) (available for evaluation in Results) where \(<\text{name}>\) is the name given to the coil in the Coil name field. The coil reactance is computed using the formula:

\[
X = \text{Im}\left(\frac{V_{\text{drop}}}{I_{\text{coil}}}\right)
\]

Inductance
This value is stored in a global variable \( \text{mf.L c cg}<\text{name}> \) (available for evaluation in Results) where \(<\text{name}><\text{name}>\) is the name given to the coil in the Coil name edit field. The coil inductance is computed using the formula:

\[
L_c = \frac{1}{\omega} \text{Im}\left(\frac{V_{\text{drop}}}{I_{\text{coil}}}\right)
\]

where \(\omega\) is the angular frequency.
Admittance
This value is stored in a global variable \( mf.Y_{cg} <\text{name}> \) (available for evaluation in Results) where \(<\text{name}>\) is the name given to the coil in the Coil name field. The coil admittance is computed from the coil impedance using the formula \( Y = Z^{-1} \).

Reversed Current Direction
Right-click the Coil Group Domain node to add the Reversed Current Direction.
To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button (\( \text{on} \)) on the Model Builder and then select the applicable option.

Show or Hide Options for Physics Feature Nodes
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\( \text{on} \)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.
See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

Domain Selection
Select the domains where you want to define the reversed current direction.

Initial Values
The Initial Values feature adds an initial value for the magnetic vector potential \( A \) that can serve as an initial value for a transient simulation or as an initial guess for a nonlinear solver.
To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button (\( \text{on} \)) on the Model Builder and then select the applicable option.

Show or Hide Options for Physics Feature Nodes
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click
the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\(\Rightarrow\)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**
Select the boundaries where you want to define an initial value.

**INITIAL VALUES**
Enter a value or expression for the initial value of the Magnetic vector potential \(\mathbf{A}\) (SI unit: Wb/m). The default 0.

*Boundary Conditions for the Magnetic Fields Interface*

With no surface currents present the interface conditions

\[
\mathbf{n}_2 \times (\mathbf{A}_1 - \mathbf{A}_2) = 0
\]
\[
\mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) = 0
\]

need to be fulfilled. Because \(\mathbf{A}\) is being solved for, the tangential component of the magnetic potential is always continuous, and thus the first condition is automatically fulfilled. The second condition is equivalent to the natural boundary condition and is hence also fulfilled unless surface currents are explicitly introduced.

*Exterior Boundaries*
These exterior boundary conditions are available:

- Magnetic Insulation (the default)
- Magnetic Field
- Surface Current
- Magnetic Potential
- Impedance Boundary Condition
- Perfect Magnetic Conductor
- Periodic Condition
- Lumped Port
Interior Boundaries
These interior boundary conditions are available:

- Magnetic Insulation
- Magnetic Potential
- Surface Current
- Perfect Magnetic Conductor
- Transition Boundary Condition
- Thin Low Permeability Gap

For axisymmetric models, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an Axial Symmetry feature to the model that is valid on the axial symmetry boundaries only.

Pairs for the Magnetic Fields Interface
These boundary conditions are available on boundary pairs. Select from the Pairs submenu.

- Sector Symmetry
- Magnetic Field
- Surface Current
- Magnetic Potential
- Perfect Magnetic Conductor
- Thin Low Permeability Gap
- Continuity

Magnetic Insulation
The Magnetic Insulation feature 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 $\mathbf{n} \times \mathbf{A} = 0$.

Magnetic insulation is a special case of the magnetic potential boundary condition that sets the tangential component of the magnetic potential to zero. It is used for the modeling of a lossless metallic surface, for example a ground plane or as a symmetry type boundary condition. It imposes symmetry for magnetic fields and “magnetic currents.” In the transient and time harmonic formulations it also imposes...
antisymmetry for electric fields and electric currents. It supports induced electric
surface currents and thus any prescribed or induced electric currents (volume, surface,
or edge currents) flowing into a perfect electric conductor boundary is automatically
balanced by induced surface currents.

The magnetic insulation boundary condition is used on exterior and interior boundaries
representing the surface of a lossless metallic conductor or (on exterior boundaries)
representing a symmetry cut. The shaded (metallic) region is not part of the model but still
carries effective mirror images of the sources. Note also that any current flowing into the
boundary is perfectly balanced by induced surface currents. The tangential vector
potential (and electric field) vanishes at the boundary.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**
For most physics interface feature nodes, the **Equation** and **Override and Contribution**
sections are displayed on a feature node **Settings** window by default. You can also click
the **Expand Sections** button on the **Model Builder** to always show some sections in an
expanded view, or go to these menus to hide options as required. Click the **Show**
button on the **Model Builder** and then select **Equation View** to display the **Equation
View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and
Expanding Advanced Feature Nodes and Sections** for more information.

**BOUNDARY SELECTION**
No user selection is required. **All boundaries** are automatically selected.
Magnetic Field

The Magnetic Field feature adds a boundary condition for specifying the tangential component of the magnetic field at the boundary:

\[ \mathbf{n} \times \mathbf{H} = \mathbf{n} \times \mathbf{H}_0 \]

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

Show or Hide Options for Physics Feature Nodes

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button \( \text{Show} \) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

Boundary Selection

Select the boundaries where you want to specify the magnetic field.

Coordinate System Selection

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

Pair Selection

If Magnetic Field is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

Magnetic Field

Enter the value or expression for the Magnetic Field \( \mathbf{H}_0 \) (SI unit: A/m) coordinates.

Surface Current

The Surface Current feature adds a boundary condition for a surface current density \( \mathbf{J}_0 \):
To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button (□) on the Model Builder and then select the applicable option.

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (□) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**
Select the boundaries where you want to specify a surface current.

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

**PAIR SELECTION**
If Surface Current is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

**SURFACE CURRENT**
Enter values or expressions for the Surface current density \( \mathbf{J}_{\text{surf}} \) (SI unit \( \text{A/m} \)) coordinates.

**Lumped Port**
Use the Lumped Port feature to apply a voltage or current excitation of a model or to connect to a circuit.

You can only apply a Lumped Port condition on boundaries that extend between two metallic boundaries—that is boundaries where Magnetic Insulation or Impedance...
conditions apply—separated by a distance much smaller than the wavelength. See S-Parameters and Ports and Lumped Ports with Voltage Input for more information.

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (Show) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

BOUNDARY SELECTION
Select the boundaries where you want to specify the lumped port.

PAIR SELECTION
If Lumped Port is selected from the Pairs menu, select the pair where you want to define this feature. First an identity pair may have to be created.

PORT PROPERTIES
Enter a unique Port Name. It is recommended to use a numeric name as it is used to define the elements of the S-parameter matrix and numeric port names are also required for port sweeps and Touchstone file export.

Type of Port
Select a Type of Port—Uniform, Coaxial, or User defined.

Select User defined for non uniform ports, for example, a curved port and enter values or expressions in the fields—Height of lumped port \( h_{\text{port}} \) (SI unit: m), Width of lumped port \( w_{\text{port}} \) (SI unit: m), and Direction between lumped port terminals \( a_{\text{h}} \).

Terminal Type
Select a Terminal type—a Cable port for a voltage driven transmission line, a Current driven port, or a Circuit port.
If **Cable** is selected, select **On** or **Off** from the **Wave excitation at this port** list to set whether it is an inport or a listener port. If **On** is selected, enter a **Voltage** $V_0$ (SI unit: V), and **Port phase** $\theta$ (SI unit: rad).

**Note:** It is only possible to excite one port at a time if the purpose is to compute S-parameters. In other cases, for example, when studying microwave heating, more than one inport might be wanted, but the S-parameter variables cannot be correctly computed so if you excite several ports, the S-parameter output is turned off.

**SETTINGS**

No entry is required if a **Circuit** terminal type is selected above.

- If a **Cable** terminal type is selected above, enter the **Characteristic impedance** $Z_{\text{ref}}$ (SI unit: $\Omega$).
- If a **Current** terminal type is selected above, enter a **Terminal current** $I_0$ (SI unit: A).

**Magnetic Potential**

The **Magnetic Potential** feature adds a boundary condition for the magnetic vector potential:

$$\mathbf{n} \times \mathbf{A} = \mathbf{n} \times \mathbf{A}_0$$

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (✓) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and Expanding Advanced Feature Nodes and Sections** for more information.
BOUNDARY SELECTION
Select the boundaries where you want to specify the magnetic potential.

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

PAIR SELECTION
If Magnetic Potential is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

MAGNETIC POTENTIAL
Enter a value or expression for the Magnetic vector potential \( A_0 \) (SI unit: Wb/m) coordinates.

Impedance Boundary Condition

The Impedance Boundary Condition provides a boundary condition that is useful at boundaries where the electromagnetic field penetrates only a short distance outside the boundary.

\[
\frac{\mu_0 \mu_r}{\eta} \mathbf{n} \times \mathbf{H} + \mathbf{E} - (\mathbf{n} \cdot \mathbf{E}) \mathbf{n} = (\mathbf{n} \cdot \mathbf{E}_s) \mathbf{n} - \mathbf{E}_s
\]

The boundary condition approximates this penetration to avoid the need to include another domain in the model. The material properties that appear in the equation are those for the domain outside the boundary.

The skin depth (that is, the distance where the electromagnetic field has decreased by a factor \( e^{-1} \)) is for a good conductor

\[
\delta = \frac{2}{\sqrt{\eta \mu_0 \varepsilon_0 \sigma}}
\]
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 $E_s$ can be used to specify a source surface current on the boundary.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (-visible) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**

Select the boundaries where you want to specify the impedance boundary condition.

**COORDINATE SYSTEM SELECTION**

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

The following material properties can be defined for the domain outside the boundary, which this boundary condition approximates. The default uses the values from material. Or select **User defined** to enter different values or expressions.

- **Relative permeability**, $\mu_r$ (unitless)
- **Relative permittivity**, $\varepsilon_r$ (unitless)
- **Electrical conductivity**, $\sigma$ (SI unit: S/m)

Enter the values of expressions for the **Source electric field** $E_s$ (SI unit: V/m) coordinates.

**Perfect Magnetic Conductor**

The **Perfect Magnetic Conductor** boundary condition $\mathbf{n} \times \mathbf{H} = 0$ is a special case of the surface current boundary condition that sets the tangential component of the magnetic field and thus also the surface current density to zero. On external boundaries, this can be interpreted as a “high surface impedance” boundary condition or used as a symmetry type boundary condition. It imposes symmetry for electric fields and electric currents. Electric currents (volume, surface, or edge currents) are not allowed to flow into a perfect magnetic conductor boundary as that would violate current conservation. On interior boundaries, the perfect magnetic conductor boundary condition literally sets the tangential magnetic field to zero which in addition to setting the surface current density to zero also makes the tangential magnetic vector potential (and in dynamics the tangential electric field) discontinuous.

The perfect magnetic conductor boundary condition is used on exterior boundaries representing the surface of a high impedance region or a symmetry cut. The shaded (high impedance) region is not part of the model but nevertheless carries effective mirror images of the sources. Note also that any electric current flowing into the boundary is forbidden as it cannot be balanced by induced electric surface currents. The tangential magnetic field vanishes at the boundary. On interior boundaries, the perfect magnetic conductor...
boundary condition literally sets the tangential magnetic field to zero which in addition to setting the surface current density to zero also makes the tangential magnetic vector potential (and in dynamics the tangential electric field) discontinuous.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\(\text{Show}\)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**

Select the boundaries that you want to model as perfect magnetic conductors.

**PAIR SELECTION**

If Perfect Magnetic Conductor is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

*Transition Boundary Condition*

The Transition Boundary Condition is used on interior boundaries to model a sheet of a medium that should be geometrically thin but it has not to be electrically thin. It represents a discontinuity in the tangential electric field. Mathematically it is described by a relation between the electric field discontinuity and the induced surface current density:
Where indices 1 and 2 refer to the different sides of the layer.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( ) on the Model Builder and then select Equation View to display the Equation node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**

Select the boundaries where you want to specify the transition boundary condition.

**TRANSITION BOUNDARY CONDITION**

The Transition Boundary Condition section contains the following material properties for the thin layer which this boundary condition approximates:

The default uses the values From material. Or select User defined to enter different values or expressions.

- **Relative permeability, \( \mu_r \) (unitless)**
- **Relative permittivity, \( \varepsilon_r \) (unitless)**
• **Electrical conductivity**, \( \sigma \) (SI unit: S/m)

• **Thickness**, \( d \) (SI unit: m)

**Thin Low Permeability Gap**

Use the **Thin Low Permeability Gap** boundary condition

\[
\mathbf{n} \times (\mathbf{H}_1 - \mathbf{H}_2) = \nabla_t \times \frac{d}{\mu_0} \nabla_t \times \mathbf{A}
\]

to model gaps filled with a material with zero conductivity such as air. This boundary condition is only applicable on interior boundaries and pair boundaries. You can also right-click to add a **Gauge-Fixing for A-Field** feature.

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

**Show or Hide Options for Physics Feature Nodes**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (\( \square \)) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and Expanding Advanced Feature Nodes and Sections** for more information.

**Boundary Selection**

Select the boundaries where you want to model a thin low permeability gap.

**Model Inputs**

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

**Coordinate System Selection**

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

The default Relative permeability $\mu_r$ (unitless) is taken from material. If User defined is selected, choose Isotropic, Diagonal, Symmetric, or Anisotropic based on the characteristics of the relative permeability, and then enter a value or expression in the field or matrix.

Enter a Surface thickness $d_s$ (SI unit: m) for the gap.

**Periodic Condition**

The Periodic Condition feature can be used to define periodicity or antiperiodicity between two boundaries. You can also activate the feature on more than two boundaries, in which case the feature tries to identify two separate surfaces which can each consist of several connected boundaries. For more complex geometries it may be necessary to use a Destination Selection subnode. This subnode makes it possible to manually specify which boundaries constitute the source and destination surfaces. To add this subnode, right-click the Periodic Condition node and select Destination Selection.

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

**Show or Hide Options for Physics Feature Nodes**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\(\text{Show}\)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder. See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Boundary Selection**

Select the boundaries where you want to apply the periodic condition.

**Periodic Condition**

Select a Type of periodicity—Continuity or Antiperiodicity. Select a Constraint type—Bidirectional, symmetric or Unidirectional. If required, select the Use weak constraints check box.
**Sector Symmetry**

Select **Sector Symmetry** at interfaces between rotating objects where sector symmetry is used. It is only available for pairs. See also **Identity and Contact Pairs** in the COMSOL Multiphysics User’s Guide.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button ( grátis ) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to **Showing and Expanding Advanced Feature Nodes and Sections** for more information.

**Boundary Selection**

Select individual boundaries in an existing identity pair. This pair first has to be created.

**Pair Selection**

When **Sector Symmetry** is selected from the **Pairs** submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

**Sector Settings**

Enter the **Number of sectors** (must be <50).

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

**Continuity**

The **Continuity** feature provides continuity in the field variables across a boundary between parts in an assembly where you have created a pair.

To display additional features for the physics interface feature nodes (and the physics interfaces), click the **Show** button ( grátis ) on the **Model Builder** and then select the applicable option.
SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (§) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

BOUNDARY SELECTION
Select individual boundaries in an existing identity pair. This pair first has to be created.

PAIR SELECTION
When Continuity is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

Line Current (Out of Plane)

The Line Current (Out of Plane) feature allows you to specify a line current out of the modeling plane. In axially symmetric geometries this is the rotational direction, in 2D geometries this is the z-direction. Select this from the Points menu.

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (§) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.
**POINT SELECTION**
Select the points where you want to add a line current.

**LINE CURRENT (OUT OF PLANE)**
Enter a value or expression for the Out of plane current \( I_0 \).

*Electric Point Dipole*

Electric point dipoles are available with 2D and 3D models. To add this feature, right-click the **Magnetic Fields** node and select **Points > Electric Point Dipole**. Apply a point dipole \( \mathbf{P} \) (SI unit: Am) to points. 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.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**
For most physics interface feature nodes, the **Equation** and **Override and Contribution** sections are displayed on a feature node **Settings** window by default. You can also click the **Expand Sections** button on the **Model Builder** to always show some sections in an expanded view, or go to these menus to hide options as required. Click the **Show** button (\( \text{Show} \)) on the **Model Builder** and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**POINT SELECTION**
Select the points where you want to add an electric point dipole.

**DIPOLE SPECIFICATION**
Select a **Dipole specification**—**Magnitude and direction** or **Dipole moment**.

**SETTINGS**
If **Magnitude and direction** is selected under **Dipole Specification**, enter coordinates for the Electric dipole moment direction \( \mathbf{n}_p \) and the Electric dipole moment, magnitude \( p \) (SI unit: Cm).
If Dipole moment is selected under Dipole Specification, enter coordinates for the Electric dipole moment \( p \) (SI unit: Cm).

**Magnetic Point Dipole**

Magnetic point dipoles are available for 2D and 3D models. To add this feature, right-click the Magnetic Fields node and select Points>Magnetic Point Dipole. Apply a point dipole \( \mathbf{M} \) (SI unit: Am\(^2\)) to points. This represents the limiting case of when the cross-section area \( a \) of a circular current loop carrying uniform current \( I \) approaches zero at while maintaining the product between \( I \) and \( a \). The dipole moment is a vector entity with the positive direction set by the curl of the current.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODERS**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\( \text{Show} \)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**POINT SELECTION**

Select the points where you want to add an electric point dipole.

**DIPOLE SPECIFICATION**

Select a Dipole specification—Magnitude and direction or Dipole moment.

**SETTINGS**

If Magnitude and direction is selected under Dipole Specification, enter coordinates for the Magnetic dipole moment direction \( \mathbf{n} \) and the Magnetic dipole moment, magnitude \( p \) (SI unit: Cm).

If Dipole moment is selected under Dipole Specification, enter coordinates for the Magnetic dipole moment \( p \) (SI unit: Cm).
The Magnetic Fields, No Currents Interface

The Magnetic Fields, No Currents interface ( ), found under the AC/DC branch ( ) of the Model Wizard, has the equations, boundary conditions, and point features for modeling magnetostatics, solving for the magnetic scalar potential. The main feature is the Magnetic Flux Conservation feature, which adds the equation for the magnetic potential and provides an interface for defining the material properties and the constitutive relation.

For a more extensive introduction to the physics and equations implemented by this interface, see the Theory for the Magnetic Fields, No Currents Interface.

When you add this interface, these default nodes are also added to the Model Builder—Magnetic Fields, No Currents, Magnetic Flux Conservation, Magnetic Insulation (the default boundary condition), and Initial Values. Right-click the Magnetic Fields, No Currents node to add other features that implement additional boundary conditions and point conditions.

**Note:** Except where described below, some of the Settings windows are the same as described for the Magnetic Fields and Electrostatics interfaces as indicated.

To display additional features for the physics interfaces and feature nodes, click the Show button ( ) in the Model Builder and select the applicable section.

**Show More Options for Physics Interfaces and Feature Nodes**

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

You can also click the Expand Sections button ( ) in the Model Builder to always show some sections or click the Show button ( ) and select Reset to Default to reset to display only the Equation and Override and Contribution sections.
For most physics feature nodes, both the **Equation** and **Override and Contribution** sections are always available. Click the **Show** button ((LED)) and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

Availability of each feature, and whether it is described for a particular interface or node, is based on the individual physics interface and feature node. For example, the **Discretization**, **Advanced Settings**, **Consistent Stabilization**, and **Inconsistent Stabilization** sections are often described individually throughout the documentation as there are unique settings. See *Showing and Expanding Advanced Feature Nodes and Sections* in the *COMSOL Multiphysics User’s Guide* for additional links to the relevant documentation.

**INTERFACE IDENTIFIER**

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

The default identifier (for the first interface in the model) is **mfnc**.

**DOMAIN SELECTION**

Select the domains where you want to define the magnetic scalar potential and the equations that describe the potential field for magnetostatics without currents. The default setting is to include all domains in the model.

**OUT-OF-PLANE THICKNESS (2D MODELS ONLY)**

Define the out-of-plane thickness $d$ by entering a value or expression (SI unit: m) in the **Thickness** edit field. The default value of 1 m is typically not representative for a thin domain. Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D models.

**DEPENDENT VARIABLES**

The dependent variable (field variable) is for the **Magnetic scalar potential** $V_m$. You can change the name in the corresponding edit field, but the names of fields and dependent variables must be unique within a model.
DISCRETIZATION
To display this section, click the Show button ( ) and select Discretization. Select Quadratic (the default), Linear, Cubic, or Quartic for the Magnetic scalar potential.

SEE ALSO
• Magnetic Flux Conservation
• Initial Values
• For Force Calculation and Infinite Elements, see The Electrostatics Interface
• Boundary Conditions for the Magnetic Fields, No Currents Interface
• Point Conditions for the Magnetic Fields, No Currents Interface

Magnetic Flux Conservation
The Magnetic Flux Conservation feature adds Equation 5-5 above (or a similar equation depending on the selected constitutive relation) for the magnetic potential and provides an interface for defining the constitutive relation and the relevant material properties (for example, the relative permeability).

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( ) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

DOMAIN SELECTION
Select the domains where you want to define the magnetic potential and the equation that describes the magnetic potential field.
MODEL INPUTS
This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty. If you add a linear temperature relation for the conductivity, you can then define the source for the temperature \( T \). From the Temperature list, select an existing temperature variable (from another physics interface) if available, or select User defined to define a value or expression for the temperature (SI unit: K) in the edit field that appears underneath the list.

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

MAGNETIC FIELD
See the settings for Magnetic Field under Ampère’s Law for the Magnetic Fields interface. Note these differences:

- Magnetic losses is not an option for this interface.
- Select BH curve (instead of HB curve) to use a curve that relates magnetic flux density \( B \) and the magnetic field \( H \) as \( |B| = f(|H|) \). To define \( |B| \), 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 edit field that appears.

Initial Values
The Initial Values feature adds an initial value for the magnetic scalar potential \( V_m \) that can serve as an initial guess for a nonlinear solver.

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\( \text{Show} \)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.
See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Domain Selection**
Select the domains where you want to define an initial value.

**Initial Values**
Enter a value or expression for the initial value of the magnetic potential $V_m$ in the Magnetic scalar potential edit field. The default value is 0 A.

*Boundary Conditions for the Magnetic Fields, No Currents Interface*

**Exterior Boundaries**
The following exterior boundary conditions are available as described in this section:

- Magnetic Flux Density
- Zero Magnetic Scalar Potential
- Magnetic Insulation (the default boundary condition)
- Magnetic Shielding

As described for the Magnetic Fields interface:

- Magnetic Potential
- Periodic Condition

In magnetostatics the relevant interface condition between two domains 1 and 2 is

$$ \mathbf{n}_2 \cdot (\mathbf{B}_1 - \mathbf{B}_2) = 0 $$

This condition provides continuity of the normal component of the magnetic flux density and is automatically satisfied by the natural boundary condition for interior boundaries, which is

$$ \mathbf{n} \cdot \{ (\mu_0 \nabla V_m - \mathbf{M})_1 - (\mu_0 \nabla V_m - \mathbf{M})_2 \} = -\mathbf{n} \cdot (\mathbf{B}_1 - \mathbf{B}_2) = 0 $$

**Interior Boundaries**
In addition, the following interior boundary conditions are available as described in this section:

- Thin Low Permeability Gap
- Magnetic Shielding
See The Magnetic Fields Interface for the following boundary conditions, which available on boundary pairs:

- Magnetic Shielding
- Sector Symmetry
- Continuity

**Magnetic Flux Density**

The Magnetic Flux Density feature adds a boundary condition for the magnetic flux density. The following equation defines the normal component of the magnetic flux density using a magnetic flux vector $B_0$:

$$ n \cdot B = n \cdot B_0 $$ \hspace{1cm} (5-1)

Using this boundary condition you can specify the normal component of the magnetic flux density at the boundary.

Alternatively, you can specify an inward (or outward) flux density using the following equation:

$$ -n \cdot B = B_n $$ \hspace{1cm} (5-2)

Using this formulation, it is possible to specify the normal component of the magnetic flux density as a scalar.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( derecho) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.
BOUNDARY SELECTION
Select the boundaries where you want to define a magnetic flux density.

MAGNETIC FLUX DENSITY
From the Type list, select which formulation of the boundary condition to use:

- Select Magnetic flux density to define the boundary condition according to Equation 5-1. You enter a value or expression for each component of the magnetic flux density $B_0$ in the corresponding edit fields.
- Select Inward flux density to define the boundary condition according to Equation 5-2. You enter a scalar value or expression for the normal component of the magnetic flux density in the $B_n$ edit field. A positive value represents an inward flux.

Zero Magnetic Scalar Potential
The Zero Magnetic Scalar Potential feature provides a boundary condition that specifies a zero magnetic potential on the boundary $V_m = 0$.

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( derecho ) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

BOUNDARY OR POINT SELECTION
Select the boundaries or points where you want to define a zero magnetic potential.
Magnetic Insulation

The Magnetic Insulation feature for the Magnetic Fields, No Currents interface provides magnetic insulation using the following boundary condition, which sets the normal component of the magnetic flux density to zero $n \cdot B = 0$.

This boundary condition is useful at boundaries confining a surrounding region of air. Magnetic insulation is the default boundary condition.

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

Show or Hide Options for Physics Feature Nodes

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (_pkt) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

Boundary Selection

Select the boundaries where you want to define magnetic insulation.

Magnetic Shielding

The Magnetic Shielding feature adds a boundary condition for magnetic shielding. It describes a thin layer of a permeable medium that shields the magnetic field. The Magnetic Shielding boundary condition uses the following equation:

$$n \cdot (B_1 - B_2) = -\nabla_t \cdot (\mu_0 \mu_r d_s \nabla_t V_m)$$

In this equation, $\nabla_t$ represents a tangential derivative (gradient), and $d_s$ is the surface thickness.

To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button (_pkt) on the Model Builder and then select the applicable option.
**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( ) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**BOUNDARY SELECTION**

Select the boundaries where you want to define magnetic shielding.

**PAIR SELECTION**

If Magnetic Shielding is selected from the Pairs > menu, select the boundary pair where you want to define this feature. First an identity pair may have to be created. Ctrl-click to deselect.

**MAGNETIC FIELD**

Define the Relative permeability \( \mu_r \). The default uses values From material. If User defined is selected, choose Isotropic, Diagonal, Symmetric, or Anisotropic and enter other values or expressions. For anisotropic material, the relative permeability is a tensor.

**THIN LAYER**

Enter a value or expression for the surface Thickness \( d_s \) (SI unit: m).

*Thin Low Permeability Gap*

The Thin Low Permeability Gap feature for the Magnetic Fields, No Current interface adds a boundary condition on an internal boundary which allows for a discontinuity in the magnetic scalar potential. You can enter a relative magnetic permeability \( \mu_r \) for the thin layer material as well as a thickness \( d_s \). The magnetic flux through this boundary is given by

\[
\mu_0 \mu_r \left( \frac{V_m^d - V_m^u}{d_s} \right)
\]

where \( V_m^u \) is the magnetic scalar potential on the upside of the boundary selection and
$V_m^\mu$ is the magnetic scalar potential on the downside.

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (√) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

PAIR SELECTION

If Thin Low Permeability Gap is selected from the Pairs submenu, select the boundary pair where you want to define this feature. First an identity pair may have to be created.

Use the thin low permeability gap boundary condition

\[
(n \cdot B)_1 = \frac{\mu_r \mu_0}{d} (V_{m1} - V_{m2})
\]

\[
(n \cdot B)_2 = \frac{\mu_r \mu_0}{d} (V_{m2} - V_{m1})
\]

to model a thin gap of a low permeable material such as air. The layer has the thickness $d$ and the relative permeability $\mu_r$.

BOUNDARY SELECTION

Select the boundaries where you want to define a thin low permeability gap.

THIN LOW PERMEABILITY GAP

Enter the surface thickness (SI unit: m) as a value or expression in the $d_s$ edit field. Also define the relative permeability using the $\mu_r$ list:

- Select From material to user the value of the relative permeability for the material.
- Select User defined to enter a value or expression for the relative permeability in the edit field that appears.
Point Conditions for the Magnetic Fields, No Currents Interface

To obtain a unique solution, you must provide the magnetic potential at one point, or at least at one point. If you use the magnetic insulation boundary condition everywhere, the potential has to be fixed using a point condition.

The following point conditions are available:

- Magnetic Potential
- Zero Magnetic Scalar Potential
The Rotating Machinery, Magnetic Interface

The Rotating Machinery, Magnetic interface ( ), found under the AC/DC branch ( ) of the Model Wizard, is available in 2D models only. It combines an out-of-plane magnetic fields (magnetic vector potential) formulation with a selection of predefined frames for prescribed rotation or rotation velocity. Thus, it shares most of its features with the magnetic fields interface; see The Magnetic Fields Interface.

In this section, only unique features of the Rotating Machinery, Magnetic interface are described. The interface only works properly if the geometry was created as an assembly pair from individual composite objects for the rotor and stator parts respectively. Also an identity pair must be defined for the rotor-stator interface boundaries under the Model/Definitions node in the model tree. See Identity and Contact Pairs in the COMSOL Multiphysics User’s Guide.

To display additional features for the physics interfaces and feature nodes, click the Show button ( ) in the Model Builder and select the applicable section.

SHOW MORE OPTIONS FOR PHYSICS INTERFACES AND FEATURE NODES

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

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

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

Availability of each feature, and whether it is described for a particular interface or node, is based on the individual physics interface and feature node. For example, the Discretization, Advanced Settings, Consistent Stabilization, and Inconsistent Stabilization sections are often described individually throughout the documentation as there are unique settings. See Showing and Expanding Advanced Feature Nodes and Sections.
in the *COMSOL Multiphysics User’s Guide* for additional links to the relevant documentation.

**INTERFACE IDENTIFIER**

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

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

**DOMAIN SELECTION**

Select the domains where you want to define. The default setting is to include all domains in the model.

**SETTINGS**

- **Out-of-plane vector** is the default and only allowed selection in the **Components** list.

**OUT-OF-PLANE THICKNESS**

Define the out-of-plane **Thickness** by entering a value or expression (SI unit: m). The default value of 1 m is typically not representative for a thin domain.

**DEPENDENT VARIABLES**

The dependent variable (field variable) is for the **Magnetic vector potential** $A$. You can change the name in the corresponding edit field, but the names of fields and dependent variables must be unique within a model.

**DISCRETIZATION**

To display this section, click the **Show** button (اظفر) and select Discretization. Select a **Magnetic vector potential**—Linear, Quadratic (the default), or Cubic.

**SEE ALSO**

- Initial Values
- Electric Field Transformation
- Prescribed Rotation
- Prescribed Rotational Velocity
- The Magnetic Fields Interface
Initial Values

The Initial Values feature adds an initial value for the magnetic vector potential.

To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button (\(\textcolor{blue}{\text{Show}}\)) on the Model Builder and then select the applicable option.

**Show or Hide Options for Physics Feature Nodes**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\(\textcolor{blue}{\text{Show}}\)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Domain Selection**

Select the domains where you want to define an initial value.

**Initial Values**

Enter a value or expression for the initial value of the Magnetic vector potential \(\mathbf{A}\).

**Electric Field Transformation**

The Electric Field Transformation feature imposes suitable transformations to the electric field definitions in all domains depending on rotational velocity.

To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button (\(\textcolor{blue}{\text{Show}}\)) on the Model Builder and then select the applicable option.

**Show or Hide Options for Physics Feature Nodes**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\(\textcolor{blue}{\text{Show}}\)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.
See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**DOMAIN SELECTION**
Select the domains where you want to apply electric field transformation. The default is all domains.

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

*Prescribed Rotation*

The Prescribed Rotation feature imposes a coordinate transformation to the selected domain that effectively rotates it a prescribed angle. It is used for the modeling of a rotating part.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (>({) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**DOMAIN SELECTION**
Select the domains where you want to use prescribed rotation.

**PRESCRIBED ROTATION**
Enter the Rotation angle (in radians) and the Rotation axis base point $r_{bp}$ components in the associated edit fields.
**Prescribed Rotational Velocity**

The *Prescribed Rotational Velocity* feature imposes a coordinate transformation to the selected domain that effectively rotates it a prescribed angle that grows linearly with time. It is used for the modeling of a rotating part.

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

**Show or Hide Options for Physics Feature Nodes**

For most physics interface feature nodes, the *Equation* and *Override and Contribution* sections are displayed on a feature node *Settings* window by default. You can also click the *Expand Sections* button on the *Model Builder* to always show some sections in an expanded view, or go to these menus to hide options as required. Click the *Show* button (\( \text{Show} \)) on the *Model Builder* and then select *Equation View* to display the *Equation View* node under all physics interface nodes in the *Model Builder*.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Domain Selection**

Select the domains where you want to use prescribed rotational velocity.

**Prescribed Rotation**

Enter the *Revolutions per second* rps (SI unit: 1/s) and the *Rotation axis base point* \( \mathbf{r}_{bp} \) components in the associated edit fields.
Theory of Magnetic and Electric Fields

Quasi-static analysis of magnetic and electric fields is valid under the assumption that \( \frac{\partial D}{\partial t} = 0 \).

In this section:

- Maxwell’s Equations
- Magnetic and Electric Potentials
- Gauge Transformations
- Selecting a Particular Gauge
- The Gauge and the Equation of Continuity for Dynamic Fields
- Explicit Gauge Fixing/Divergence Constraint
- Ungauged Formulations and Current Conservation
- Time-Harmonic Magnetic Fields

Maxwell’s Equations

This implies that it is possible to rewrite Maxwell’s equations in the following manner:

\[
\nabla \times \mathbf{H} = \mathbf{J} = \sigma (\mathbf{E} + \mathbf{v} \times \mathbf{B}) + \mathbf{J}_e
\]
\[
\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t}
\]
\[
\nabla \cdot \mathbf{B} = 0
\]
\[
\nabla \cdot \mathbf{D} = \rho
\]
\[
\nabla \cdot \mathbf{J} = 0
\]

Here \( \mathbf{J}_e \) is an externally generated current density and \( \mathbf{v} \) is the velocity of the conductor. The crucial criterion for the quasi-static approximation to be valid is that the currents and the electromagnetic fields vary slowly. This means that the dimensions of the structure in the problem need to be small compared to the wavelength.

See also

- Magnetic and Electric Potentials
- Gauge Transformations
• Selecting a Particular Gauge
• The Gauge and the Equation of Continuity for Dynamic Fields
• Explicit Gauge Fixing/Divergence Constraint
• Ungauged Formulations and Current Conservation
• Time-Harmonic Magnetic Fields

_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
\[ \sigma \frac{\partial \mathbf{A}}{\partial t} + \nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \nabla \times (\nabla \times \mathbf{A}) + \sigma \nabla V = \mathbf{J}_e \]

Equation 5-3

The equation of continuity, which is obtained by taking the divergence of the above equation, adds the following equation:
\[ \nabla \cdot \left( -\sigma \frac{\partial \mathbf{A}}{\partial t} + \sigma \nabla \times (\nabla \times \mathbf{A}) - \sigma \nabla V + \mathbf{J}_e \right) = 0 \]

Equation 5-4

Equation 5-3 and Equation 5-4 form a system of equations for the two potentials \( \mathbf{A} \) and \( V \).

_SEE ALSO_
• Maxwell’s Equations
• Gauge Transformations
• Selecting a Particular Gauge
• The Gauge and the Equation of Continuity for Dynamic Fields
• Explicit Gauge Fixing/Divergence Constraint
• Ungauged Formulations and Current Conservation
• Time-Harmonic Magnetic Fields
Gauge Transformations

The electric and magnetic potentials are not uniquely defined from the electric and magnetic fields through

\[
E = -\frac{\partial A}{\partial t} - \nabla V
\]

\[
B = \nabla \times A
\]

Introducing two new potentials

\[
\tilde{A} = A + \nabla \Psi
\]

\[
\tilde{V} = V - \frac{\partial \Psi}{\partial t}
\]

gives the same electric and magnetic fields:

\[
E = -\frac{\partial \tilde{A}}{\partial t} - \nabla \tilde{V} = -\frac{\partial (A + \nabla \Psi)}{\partial t} - \nabla (V - \frac{\partial \Psi}{\partial t}) = -\frac{\partial \tilde{A}}{\partial t} - \nabla \tilde{V}
\]

\[
B = \nabla \times \tilde{A} = \nabla \times (A + \nabla \Psi) = \nabla \times \tilde{A}
\]

The variable transformation of the potentials is called a \textit{gauge transformation}. To obtain a unique solution you need to choose the gauge, that is, put constraints on \(\Psi\) that make the solution unique. Another way of expressing this additional condition is to put a constraint on \(\nabla \cdot A\). A vector field is uniquely defined up to a constant if both \(\nabla \cdot A\) and \(\nabla \times A\) are given. This is called \textit{Helmholtz's theorem}.

One particular gauge is the \textit{Coulomb gauge} given by the constraint: \(\nabla \cdot A = 0\).

\textbf{See Also}
- Maxwell’s Equations
- Magnetic and Electric Potentials
- Selecting a Particular Gauge
- The Gauge and the Equation of Continuity for Dynamic Fields
- Explicit Gauge Fixing/Divergence Constraint
- Ungauged Formulations and Current Conservation
- Time-Harmonic Magnetic Fields
**Selecting a Particular Gauge**

Important observations are that in the dynamic case $A$ and $V$ are coupled via the selected gauge. For a dynamic formulation, it is also possible to select a $\Psi$ 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 $A$. The Magnetic and Electric fields interface in the AC/DC Module involves both $A$ and $V$ and is inherently ungauged for all study types. In the static limit, $A$ and $V$ are not coupled via the gauge selection and thus any gauge can be chosen for $A$ when performing magnetostatic modeling.

**SEE ALSO**
- Maxwell’s Equations
- Magnetic and Electric Potentials
- Gauge Transformations
- The Gauge and the Equation of Continuity for Dynamic Fields
- Explicit Gauge Fixing/Divergence Constraint
- Ungauged Formulations and Current Conservation
- Time-Harmonic Magnetic Fields

**The Gauge and the Equation of Continuity for Dynamic Fields**

After eliminating the electric potential by choosing the appropriate gauge and disregarding the velocity term. The equation of continuity obtained by taking the divergence of Ampère’s law reads:

$$\nabla \cdot \left( -\sigma \frac{\partial A}{\partial t} + J_e \right) = 0$$

It is clear that unless the electrical conductivity is uniform, the particular gauge used to eliminate $V$ cannot be the Coulomb gauge as that would violate the equation of continuity and would thereby also violate Ampère’s law.

**SEE ALSO**
- Maxwell’s Equations
- Magnetic and Electric Potentials
- Gauge Transformations
• Selecting a Particular Gauge
• Explicit Gauge Fixing/Divergence Constraint
• Ungauged Formulations and Current Conservation
• Time-Harmonic Magnetic Fields

Explicit Gauge Fixing/Divergence Constraint

The AC/DC Module has a gauge fixing feature that is imposed by adding an extra scalar field variable $\psi$ (not to be confused with $\Psi$ used in the gauge transformation in the preceding section). The $\psi$ 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 $\psi$ is used to impose the Coulomb gauge: $\nabla \cdot \mathbf{A} = 0$. However, to get a closed set of equations, $\psi$ must be able to affect the first equation and this is obtained by modifying the first equation to:

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

The additional term on the right-hand side can be seen as a Lagrange multiplier that not only imposes the Coulomb gauge but also eliminates any divergence in the externally generated current density, $\mathbf{J}_e$ and makes it comply with the current continuity inherent in Ampère’s law.

The gauge fixing feature similarly imposes the Coulomb gauge also for the dynamic (frequency domain) study type in the Magnetic and Electric Fields interface.

For the dynamic (frequency domain and time dependent) study types for the Magnetic Fields interface the gauge is already determined so the gauge fixing feature is not allowed to impose the Coulomb gauge but reduces to help imposing current conservation. The first one is for the frequency domain study and the second one is for the time dependent study type.

$$\nabla \cdot \mathbf{J} = 0$$
$$\nabla \cdot (\sigma \mathbf{A}) = 0$$
The main benefit of using this kind of divergence constraint is improved numerical stability, especially when approaching the static limit when the inherent gauge deteriorates.

**SEE ALSO**
- Maxwell’s Equations
- Magnetic and Electric Potentials
- Gauge Transformations
- Selecting a Particular Gauge
- The Gauge and the Equation of Continuity for Dynamic Fields
- Ungauged Formulations and Current Conservation
- Time-Harmonic Magnetic Fields

**Ungauged Formulations and Current Conservation**

Current conservation is inherent in Ampère’s law and it is known that if current is conserved, explicit gauge fixing is not necessary as iterative solvers converge towards a valid solution. However, it is generally not sufficient for the source currents to be divergence free in an analytical sense as when interpolated on the finite element functional basis, this property is not conserved.

When using the Magnetic and Electric Fields interface the electric potential is used to state current conservation so unless nonphysical current sources are specified inside the computational domain current conservation is fulfilled.

When using the Magnetic Fields interface, current conservation is usually imposed either by the solver (for magnetostatics) or in the transient or time harmonic case by the induced current density. The explicit gauge or divergence constraint can also help imposing current conservation as described in the preceding section.

**SEE ALSO**
- Maxwell’s Equations
- Magnetic and Electric Potentials
- Gauge Transformations
- Selecting a Particular Gauge
- The Gauge and the Equation of Continuity for Dynamic Fields
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):

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

In the transient case the inclusion of this term would lead to a second-order equation in time, but in the harmonic case there are no such complications. Using the definition of the electric and magnetic potentials, the system of equations becomes

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

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

The constitutive relation $\mathbf{D} = \varepsilon_0 \mathbf{E} + \mathbf{P}$ has been used for the electric field.

You obtain a particular gauge that reduces the system of equation by choosing $\Psi = -j \mathbf{V}/\omega$ in the gauge transformation. This gives

$$\tilde{\mathbf{A}} = \mathbf{A} - \frac{j}{\omega} \nabla \mathbf{V}$$

$$\tilde{\mathbf{V}} = 0$$

When $\tilde{\mathbf{V}}$ vanishes from the equations, you only need the second one,

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

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.

SEE ALSO

- Maxwell’s Equations
- Magnetic and Electric Potentials
- Gauge Transformations
- Selecting a Particular Gauge
- The Gauge and the Equation of Continuity for Dynamic Fields
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.

You can use the Magnetic Fields interface for 3D, 2D in-plane, and 2D axisymmetric models. Unless you have a license for the AC/DC Module, only 2D modeling involving out-of-plane currents and axisymmetric modeling involving azimuthal currents are supported. For a deeper theoretical background to the magnetic vector potential used below, see the section starting with Maxwell’s Equations.

In this section:

- Magnetostatics Equation
- Frequency Domain Equation
- Transient Equation

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, $\mathbf{B} = \mu_0 (\mathbf{H} + \mathbf{M})$, you can 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 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 axisymmetric formulations.

**Transient Equation**

The transient equation this physics interface solves is Ampère’s law, here illustrated with the constitutive relation \( \mathbf{B} = \mu_0 (\mathbf{H} + \mathbf{M}) \). The term involving the velocity only applies in the 2D and axisymmetric formulations.

\[ \frac{\partial^2 \mathbf{A}}{\partial t^2} + \nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) = \mathbf{J}_e \]
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 you can define the magnetic scalar potential $V_m$ 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

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

(5-5)

The Magnetic Fields, No Currents interface uses this equation for modeling of magnetostatics in the absence of electric currents.

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

$$-\nabla \cdot d(\mu_0 \nabla V_m - \mu_0 \mathbf{M}) = 0$$
The Magnetic and Electric Fields Interface

In this chapter the Magnetic and Electric fields interface, found under the AC/DC branch (💧) in the Model Wizard, is described.

- The Magnetic and Electric Fields Interface
- Theory for the Magnetic and Electric Fields Interface
The Magnetic and Electric Fields Interface

The Magnetic and Electric Fields interface (\(\text{\textcopyright} \)), found under the AC/DC branch (\(\text{\textcopyright} \)) of the Model Wizard, has the equations, boundary conditions, and external currents for modeling electric and magnetic fields, solving for the electric potential and magnetic vector potential.

For a more thorough introduction to the equations solved by this physics interface, see the Theory for the Magnetic and Electric Fields Interface.

The main feature is the Ampère’s Law and Current Conservation feature, which adds the equation for the electric potential and magnetic vector potential and provides an interface for defining the constitutive relations and their associated properties such as the relative permeability, relative permittivity, and electrical conductivity.

When you add this interface, these default nodes are also added to the Model Builder—Ampère’s Law and Current Conservation, Magnetic Insulation (the default boundary condition for the magnetic vector potential), and Initial Values.

Right-click the Magnetic and Electric Fields node to add other features that implement, for example, boundary conditions and external currents.

**Note:** Except where described below, the majority of the Settings windows are the same as for the Magnetic Fields, Electrostatics, and Electric Currents interfaces as indicated.

To display additional features for the physics interfaces and feature nodes, click the Show button (\(\text{\textcopyright} \)) in the Model Builder and select the applicable section.

**Show more options for physics interfaces and feature nodes**

After clicking the Show button (\(\text{\textcopyright} \)), some sections display on the Settings window when a node is clicked and other features are available from the context menu when a node is right-clicked. For each physics interface, the additional sections that can be displayed included Equation, Advanced Settings, Discretization, Consistent Stabilization, and Inconsistent Stabilization.
You can also click the Expand Sections button (.expand_more) in the Model Builder to always show some sections or click the Show button (show) and select Reset to Default to reset to display only the Equation and Override and Contribution sections.

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

Availability of each feature, and whether it is described for a particular interface or node, is based on the individual physics interface and feature node. For example, the Discretization, Advanced Settings, Consistent Stabilization, and Inconsistent Stabilization sections are often described individually throughout the documentation as there are unique settings. See Showing and Expanding Advanced Feature Nodes and Sections in the COMSOL Multiphysics User’s Guide for additional links to the relevant documentation.

INTERFACE IDENTIFIER

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

The default identifier (for the first interface in the model) is \texttt{mef}.

DOMAIN SELECTION

Select the domains where you want to define the electric potential, the magnetic vector potential, and the equations that describe the Magnetic and Electric Fields. The default setting is to include all domains in the model.

SETTINGS (2D MODELS ONLY)

Select Components—Out-of-plane vector (the default), In-plane vector, or Three-component vector for the magnetic vector potential. From the practical viewpoint this choice is equivalent to deciding in what directions the electric current is allowed to flow (out-of-plane currents, in-plane currents or currents flowing in all three coordinate directions).
**OUT-OF-PLANE THICKNESS (2D MODELS ONLY)**

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

**PORT SWEEP SETTINGS**

Select the **Activate port sweep** check box to invoke a parametric sweep over the ports/terminals in addition to the automatically generated frequency sweep. The generated lumped parameters are in the form of an impedance or admittance matrix depending on the port/terminal settings which consistently must be of either fixed voltage or fixed current type. The **Port parameter name** input field assigns a specific name to the variable that controls the port number solved for during the sweep. The lumped parameters are subject to **Touchstone file export**. File name and path are entered in an input field. See also **Lumped Parameters** for more information.

*Solving for a Port Sweep*

An additional step is required when solving for a port sweep. You need to right-click the **Study** node in the model tree and add a **Parametric Sweep**. In the new **Parametric Sweep** node, you enter for **Parameter names**, the **Port parameter name** specified when activating the port sweep and for **Parameter values**, you enter the desired list with terminal numbers. You then need to right-click the **Study** node and select **Other** and **Generate Sequences from Study** before solving.

**DEPENDENT VARIABLES**

The dependent variables (field variables) are for the **Electric potential** \( V \) and **Magnetic vector potential** \( \mathbf{A} \). You can change the name in the corresponding edit field, but the names of fields and dependent variables must be unique within a model.

**DISCRETIZATION**

To display this section, click the **Show** button (\( \text{Show} \)) and select **Discretization**. Select **Quadratic** (the default), **Linear**, **Cubic**, or **Quartic** for the **Electric potential** and **Magnetic vector potential**.

**SEE ALSO**

- Ampère’s Law and Current Conservation
- Initial Values
- Domain Conditions for the Magnetic and Electric Fields Interface
- Boundary Conditions for the Magnetic and Electric Fields Interface
Point and Edge Conditions for the Magnetic and Electric Fields Interface

Ampère’s Law and Current Conservation

The Ampère’s Law and Current Conservation node adds Ampère’s law and the equation of continuity. It provides an interface for defining the constitutive relations and their associated properties as described in the Theory for the Magnetic and Electric Fields Interface.

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

Show or Hide Options for Physics Feature Nodes

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( ) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

Domain Selection

Select the domains where you want to define the magnetic vector potential and the equation based on Ampère’s law that defines the potential.

Model Inputs

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty. If you add a linear temperature relation for the conductivity, you can then define the source for the temperature $T$. From the Temperature list, select an existing temperature variable (from another physics interface) if available, or select User defined to define a value or expression for the temperature (SI unit: K) in the edit field that appears underneath the list.

Coordinate System Selection

The Global coordinate system is selected by default. The Coordinate system list contains any additional coordinate systems that the model includes.
CONDUCTION CURRENT
See the settings for Conduction Current under Ampère’s Law for the Magnetic Fields interface.

ELECTRIC FIELD
See the settings for Electric Field under Charge Conservation for the Electrostatics interface.

MAGNETIC FIELD
See the settings for Magnetic Field under Ampère’s Law for the Magnetic Fields interface.

Initial Values
The Initial Values feature adds an initial value for the magnetic vector potential and electric potential that can serve as an initial value for a transient simulation or as an initial guess for a nonlinear solver.

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button ( ) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

BOUNDARY SELECTION
Select the boundaries where you want to define an initial value.

INITIAL VALUES
Enter a value or expression for the initial value of the Magnetic vector potential \( A \) (SI unit: Wb/m) and initial value of the Electric potential \( V \) (SI unit: V). The default values are 0.
Domain Conditions for the Magnetic and Electric Fields Interface

Except for Ampère’s Law and Current Conservation and Initial Values the other Settings windows are the same as for the Magnetic Fields, Electrostatics, and Electric Currents interfaces as indicated.

See The Magnetic Fields Interface for these features:

- Ampere’s Law with a default Gauge Fixing for A-Field node
- Velocity (Lorentz Term)
- Multi-Turn Coil Domain (2D and 2D axisymmetric models)
- Single-Turn Coil Domain (2D and 2D axisymmetric models)
- Coil Group Domain (2D and 2D axisymmetric models)

See The Electrostatics Interface for these features:

- Force Calculation
- Infinite Elements. A default Ampere’s Law and Current Conservation feature is added. Also right-click to add Charge Conservation (described for the Electrostatics interface) or Ampere’s Law features.

See The Electric Currents Interface for these features:

- External Current Density

Boundary Conditions for the Magnetic and Electric Fields Interface

In the Magnetic and Electric Fields Interface, the boundary conditions are applied in a two-step procedure. The reason is that only certain combinations of electric and magnetic boundary conditions are physically relevant whereas others may lead to nonphysical models violating current conservation. The procedure is to first select the magnetic boundary conditions. Those are available directly by right-clicking the physics interface node. In a second step the electric boundary conditions are selected, as subfeatures to the magnetic ones, by right-clicking each of the magnetic boundary condition feature nodes. For each boundary feature or subfeature all of the Settings windows are the same as for the Magnetic Fields, Electrostatics, and Electric Currents interfaces as indicated.

Magnetic Boundary Conditions

With no surface currents present the interface conditions
need to be fulfilled. Because the physics interface solves for $\mathbf{A}$, the tangential component of the magnetic potential is always continuous, and thus the first condition is automatically fulfilled. The second condition is equivalent to the natural boundary condition and is hence also fulfilled unless surface currents are explicitly introduced.

See The Magnetic Fields Interface for all of the following features:

**Exterior Boundaries**
- Magnetic Insulation (the default boundary condition)

**Note:** For the Magnetic and Electric Fields interface, a default Electric Insulation feature is also added to Magnetic Insulation. You can also right-click to add other nodes.

- Surface Current
- Impedance Boundary Condition
- Magnetic Field
- Magnetic Potential
- Perfect Magnetic Conductor

**Interior Boundaries**
- Magnetic Insulation
- Surface Current
- Magnetic Potential
- Transition Boundary Condition
- Lumped Port (also for pairs)
- Thin Low Permeability Gap

See The Electric Currents Interface for the following feature:
- Contact Resistance

For axisymmetric models, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an Axial Symmetry feature to the model that is valid on the axial symmetry boundaries only.
**ELECTRIC BOUNDARY CONDITIONS**

The relevant interface condition at interfaces between different media and interior boundaries is continuity; that is,

\[ \mathbf{n}_2 \cdot (\mathbf{J}_1 - \mathbf{J}_2) = 0 \]

which is the natural boundary condition.

**Exterior Boundaries**

When you right-click a magnetic boundary condition feature, some of the following are available for the electric exterior boundary conditions as described for the interface indicated:

See [The Electrostatics Interface](#) for these features:

- Electric Potential - also available for points (2D and 3D)
- Ground - also available for points (2D and 3D)
- Floating Potential
- Terminal
- Periodic Condition - right-click to add a Destination Selection feature

See [The Electric Currents Interface](#) for these features:

- Electric Insulation - this feature is added to the Magnetic Insulation feature by default
- Normal Current Density

**Interior Boundaries**

The interior electric boundary conditions that are available depend on the magnetic boundary condition that has been selected. See [The Electric Currents Interface](#) for these features:

- Ground
- Electric Potential
- Terminal

For axisymmetric models, COMSOL Multiphysics takes the axial symmetry boundaries (at \( r = 0 \)) into account and automatically adds an *Axial Symmetry* feature to the model that is valid on the axial symmetry boundaries only.
GENERAL BOUNDARY CONDITIONS
Periodic Condition, Sector Symmetry, and Continuity are all available. See The Magnetic Fields Interface.

Point and Edge Conditions for the Magnetic and Electric Fields Interface
All of the Settings windows (except for Edge Current) are the same as for other interfaces as indicated.

- Edge Current
- Magnetic Point Dipole and Electric Point Dipole—see The Magnetic Fields Interface

Edge Current
The Edge Current feature allows you to specify a line current along one or more edges. To display additional features for the physics interface feature nodes (and the physics interfaces), click the Show button (✓) on the Model Builder and then select the applicable option.

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (✓) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

EDGE SELECTION
Select the edges that you want to an edge current.

EDGE CURRENT
Specify the Edge current $I_0$ (SI unit: A).
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. You can use this physics interface for 3D, 2D in-plane, and 2D axisymmetric models. For a deeper theoretical background to the magnetic vector potential and electric scalar potential used below, please refer to the section Theory for the Magnetic and Electric Fields Interface. Note that the Magnetic and Electric Currents interface only supports the stationary and frequency domain study types—that is, there is no transient formulation available.

In this section:
- Magnetostatics Equations
- Frequency Domain Equations

Magnetostatics Equations

To derive the magnetostatics equations, start with Ampère’s law for static cases:
\( \nabla \times \mathbf{H} = \mathbf{J} \). Define the potentials,
\[
\mathbf{B} = \nabla \times \mathbf{A} \\
\mathbf{E} = -\nabla \mathbf{V}
\]
and the current
\[
\mathbf{J} = \sigma \mathbf{v} \times \mathbf{B} - \sigma \nabla \mathbf{V} + \mathbf{J}^e
\]
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 you can rewrite Ampère’s law as
\[
\nabla \times (\mu_0^{-1} \nabla \times \mathbf{A} - \mathbf{M}) - \sigma \mathbf{v} \times (\nabla \times \mathbf{A}) + \sigma \nabla \mathbf{V} = \mathbf{J}^e
\]
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 \( \mathbf{V} \) and \( \mathbf{A} \) apply:
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 and use the definitions of the fields, and combine them with the constitutive relationships to rewrite Ampère’s law as

\[ \mathbf{B} = \nabla \times \mathbf{A} \]
\[ \mathbf{E} = -\nabla \mathbf{v} - j\omega \mathbf{A} \]

and use the definitions of the fields,

\[ \mathbf{B} = \nabla \times \mathbf{A} \]
\[ \mathbf{E} = -\nabla \nabla \mathbf{v} - j\omega \mathbf{A} \]

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

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

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 \( A \) apply:

\[ -\nabla \cdot ((j\omega\sigma - \omega^2 \varepsilon_0)\mathbf{A} - \sigma \nabla \times (\nabla \times \mathbf{A}) + (\sigma + j\omega \varepsilon_0)\nabla V - (\mathbf{J}_e + j\omega \mathbf{P})) = 0 \]
\[ (j\omega\sigma - \omega^2 \varepsilon_0)\mathbf{A} + \nabla \times (\mu_0^{-1}\nabla \times \mathbf{A} - \mathbf{M}) - \sigma \nabla \times (\nabla \times \mathbf{A}) + (\sigma + j\omega \varepsilon_0)\nabla V - j\omega \mathbf{P} = \mathbf{J}_e \]

**SEE ALSO**
- Frequency Domain Equations
- Magnetostatics Equations
The Electrical Circuit Interface

The Electrical Circuit interface, found under the AC/DC branch in the Model Wizard, has the equations for modeling electrical circuits with or without connections to a distributed fields model, solving for the voltages, currents and charges associated with the circuit elements.

In this chapter:

- The Electrical Circuit Interface
- Theory for the Electrical Circuit Interface
The Electrical Circuit Interface

The Electrical Circuit interface ( ), found under the AC/DC branch ( ) in the Model Wizard, has the equations for modeling electrical circuits with or without connections to a distributed fields model, solving for the voltages, currents and charges associated with the circuit elements.

For more details on the equations solved by this physics interface, see the Theory for the Electrical Circuit Interface.

When you add this interface, it adds a default Ground Node feature and associates that with node zero in the electrical circuit.

Note: Circuit nodes are nodes in the electrical circuit and should not be confused with nodes in the model tree of COMSOL Multiphysics. Circuit node names are not restricted to numerical values but can be arbitrary character strings.

To display additional features for the physics interfaces and feature nodes, click the Show button ( ) in the Model Builder and select the applicable section.

Show more options for physics interfaces and feature nodes

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

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

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

Availability of each feature, and whether it is described for a particular interface or node, is based on the individual physics interface and feature node. For example, the Discretization, Advanced Settings, Consistent Stabilization, and Inconsistent Stabilization
sections are often described individually throughout the documentation as there are unique settings. See Showing and Expanding Advanced Feature Nodes and Sections in the COMSOL Multiphysics User’s Guide for additional links to the relevant documentation.

**INTERFACE IDENTIFIER**

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

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

**SEE ALSO**

- Ground Node
- Resistor
- Capacitor
- Inductor
- Voltage Source
- Current Source
- Voltage-Controlled Voltage Source
- Voltage-Controlled Current Source
- Current-Controlled Voltage Source
- Current-Controlled Current Source
- Subcircuit Definition
- Subcircuit Instance
- NPN BJT
- n-Channel MOSFET
- Diode
- External I vs. U
- External U vs. I
- External I-Terminal
- SPICE Circuit Import
**Ground Node**

The **Ground Node** node (\(\text{\(\square\)}\)) adds a ground node with the default node number zero to the electrical circuit. This is the default feature in the Electrical Circuit interface.

**GROUND CONNECTION**

Set the **Node name** for the ground node in the circuit. The convention is to use zero for the ground node.

**Resistor**

The **Resistor** node (\(\text{\(\square\)}\)) connects a resistor between two nodes in the electrical circuit.

**NODE CONNECTIONS**

Set the two **Node names** for the connecting nodes for the resistor. If the ground node is involved, the convention is to use zero for this.

**DEVICE PARAMETERS**

Enter the **Resistance** of the resistor.

**Capacitor**

The **Capacitor** node (\(\text{\(\square\)}\)) connects a capacitor between two nodes in the electrical circuit.

**NODE CONNECTIONS**

Set the two **Node names** for the connecting nodes for the capacitor. If the ground node is involved, the convention is to use zero for this.

**DEVICE PARAMETERS**

Enter the **Capacitance** of the capacitor.

**Inductor**

The **Inductor** node (\(\text{\(\square\)}\)) connects an inductor between two nodes in the electrical circuit.

**NODE CONNECTIONS**

Set the two **Node names** for the connecting nodes for the inductor. If the ground node is involved, the convention is to use zero for this.
DEVICE PARAMETERS
Enter the Inductance of the inductor.

Voltage Source

The Voltage Source node (\(\circ\)) connects a voltage source between two nodes in the electrical circuit.

NODE CONNECTIONS
Set the two Node names for the connecting nodes for the voltage source. The first node represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS
Enter the Source type that should be adapted to the selected study type. It can be DC-source, AC-source, or a time-dependent Sine source. Depending on the choice of source, you also specify the Voltage, \(V_{\text{src}}\), the offset Voltage, \(V_{\text{off}}\), the Frequency, and the Source phase. All values are peak values rather than RMS. Note that for the AC source, the frequency is a global input set by the solver so do not use the Sine source unless the model is time dependent.

Current Source

The Current Source node (\(\circ\)) connects a current source between two nodes in the electrical circuit.

NODE CONNECTIONS
Set the two Node names for the connecting nodes for the current source. The first node represents the positive reference terminal from which the current flows through the source to the second node. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS
Enter the Source type which should be adapted to the selected study type. It can be DC-source, AC-source or a time-dependent Sine source. Depending on the choice of source, you also specify the Current, \(I_{\text{src}}\), the offset Current, \(I_{\text{off}}\), the Frequency and the Source phase. All values are peak values rather than RMS. Note that for the AC source, the frequency is a global input set by the solver so do not use the Sine source unless the model is time-dependent.
**Voltage-Controlled Voltage Source**

The Voltage-Controlled Voltage Source node (\(\text{\(1\)}\)) connects a voltage-controlled voltage source between two nodes in the electrical circuit. A second pair of nodes define the input control voltage.

**NODE CONNECTIONS**

Specify four Node names: the first pair for the connection nodes for the voltage source and the second pair defining the input control voltage. The first node in a pair represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this.

**DEVICE PARAMETERS**

Enter the voltage Gain. The resulting voltage is this number multiplied by the control voltage.

---

**Voltage-Controlled Current Source**

The Voltage-Controlled Current Source node (\(\text{\(1\)}\)) connects a voltage-controlled current source between two nodes in the electrical circuit. A second pair of nodes define the input control voltage.

**NODE CONNECTIONS**

Specify four Node names: the first pair for the connection nodes for the current source and the second pair defining the input control voltage. The first node in a pair represents the positive voltage reference terminal or the one from which the current flows through the source to the second node. If the ground node is involved, the convention is to use zero for this.

**DEVICE PARAMETERS**

Enter the voltage Gain. The resulting current is this number multiplied by the control voltage. Thus it formally has the unit of conductance.

---

**Current-Controlled Voltage Source**

The Current-Controlled Voltage Source node (\(\text{\(1\)}\)) connects a current-controlled voltage source between two nodes in the electrical circuit. The input control current is the one flowing through a named device that must be a two-pin device.
**NODE CONNECTIONS**

Set two Node names for the connection nodes for the voltage source. The first node in a pair represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this.

**DEVICE PARAMETERS**

Enter the voltage Gain and the Device (any two-pin device) name. The resulting voltage is this number multiplied by the control current through the named Device (any two-pin device). Thus it formally has the unit of resistance.

**Current-Controlled Current Source**

The Current-Controlled Current Source node ( ) connects a current-controlled current source between two nodes in the electrical circuit. The input control current is the one flowing through a named device that must be a two-pin device.

**NODE CONNECTIONS**

Specify two Node names for the connection nodes for the current source. The first node in a pair represents the positive reference terminal from which the current flows through the source to the second node. If the ground node is involved, the convention is to use zero for this.

**DEVICE PARAMETERS**

Enter the current Gain and the Device (any two-pin device) name. The resulting current is this number multiplied by the control current through the named Device (any two-pin device).

**Subcircuit Definition**

The Subcircuit Definition node ( ) is used to define subcircuits. By right-clicking a subcircuit definition node, you can add all circuit features available except for the subcircuit definition feature itself. By right-clicking a subcircuit definition node, you can also Rename it to something more descriptive than its default name.

**SUBCIRCUIT PINS**

Define the Pin names at which the subcircuit connects to the main circuit or to other subcircuits when referenced by a Subcircuit Instance feature. The Pin names refer to circuit nodes in the subcircuit. The order in which the Pin names are defined is the order in which they are referenced by a Subcircuit Instance feature.
Subcircuit Instance

The **Subcircuit Instance** node ([±]) is used to refer to defined subcircuits.

**NODE CONNECTIONS**

Select the **Name of subcircuit link** from the list of defined subcircuits in the circuit model and the circuit **Node names** at which the subcircuit instance connects to the main circuit or to another subcircuit if used therein.

**NPN BJT**

The **NPN BJT** device model ([ґ]) is a large-signal model for an NPN bipolar junction transistor (BJT). It is an advanced device model and no thorough description and motivation of the many input parameters is attempted here. The interested reader is referred to Ref. 2 for more details on semiconductor modeling within circuits. Many device manufacturers provide model input parameters for this BJT model. For any particular make of BJT, the device manufacturer should be the primary source of information.

**NODE CONNECTIONS**

Specify three **Node names** for the connection nodes for the **NPN BJT** device. These represent the collector, base, and emitter nodes, respectively. If the ground node is involved, the convention is to use zero for this.

**MODEL PARAMETERS**

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular BJT, the device manufacturer should be the primary source of information. For an explanation of the **Model Parameters** see NPN Bipolar Transistor.

**n-Channel MOSFET**

The **n-Channel MOSFET** device model ([ґ]) is a large-signal model for an n-Channel MOS transistor (MOSFET). It is an advanced device model and no thorough description and motivation of the many input parameters is attempted here. The interested reader is referred to Ref. 2 for more details on semiconductor modeling within circuits. Many device manufacturers provide model parameters for this MOSFET model. For any particular make of MOSFET, the device manufacturer should be the primary source of information.
NODE CONNECTIONS
Specify four Node names for the connection nodes for the n-Channel MOSFET device. These represent the drain, gate, source, and bulk nodes, respectively. If the ground node is involved, the convention is to use zero for this.

MODEL PARAMETERS
Specify the Model Parameters. Reasonable defaults are provided but for any particular MOSFET, the device manufacturer should be the primary source of information. For an explanation of the Model Parameters see n-Channel MOS Transistor.

Diode
The Diode device model is a large-signal model for a diode. It is an advanced device model and no thorough description and motivation of the many input parameters is attempted here. The interested reader is referred to Ref. 2 for more details on semiconductor modeling within circuits. Many device manufacturers provide model parameters for this diode model. For any particular make of diode, the device manufacturer should be the primary source of information.

NODE CONNECTIONS
Specify two Node names for the positive and negative nodes for the Diode device. If the ground node is involved, the convention is to use zero for this.

MODEL PARAMETERS
Specify the Model Parameters. Reasonable defaults are provided but for any particular diode, the device manufacturer should be the primary source of information. For an explanation of the Model Parameters see Diode.

External I vs. U
The External I vs. U node connects an arbitrary voltage measurement, for example a circuit terminal or circuit port boundary or a coil domain from another physics interface, as a source between two nodes in the electrical circuit. The resulting circuit current from the first node to the second node is typically coupled back as a prescribed current source in the context of the voltage measurement.
**NODE CONNECTIONS**
Specify the two Node names for the connecting nodes for the voltage source. The first node represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this.

**EXTERNAL DEVICE**
Enter the source of the Voltage. If you have circuit or current excited terminals or circuit ports defined on boundaries or a multiturn coil domains in other physics interfaces, these display as options in the Voltage list. You can also select the User defined option and type in your own voltage variable, for example, using a suitable coupling operator (see Model Couplings in the COMSOL Multiphysics User’s Guide). For inductive or electromagnetic wave propagation models, the voltage measurement must be performed as an integral of the electric field as the electric potential only does not capture induced EMF. Also the integration must be performed over a distance that is short compared to the local wavelength. Note that except for when coupling to a circuit terminal or circuit port, you must manually couple back the current flow variable in the electrical circuit to the context of the voltage measurement. This applies also when coupling to a current excited terminal. The name of this current variable follows the convention cirn.IvsUm_i, where cirn is the tag of the Electrical Circuit interface node and IvsUm is the tag of the External I vs. U node. The mentioned tags are typically displayed within curly braces {} in the model tree.

*External U vs. I*

The External U vs. I node connects an arbitrary current measurement, for example, from another physics interface, as a source between two nodes in the electrical circuit. The resulting circuit voltage between the first node and the second node is typically coupled back as a prescribed voltage source in the context of the current measurement.

**NODE CONNECTIONS**
Specify the two Node names for the connecting nodes for the current source. The current flows from the first node to the second node. If the ground node is involved, the convention is to use zero for this.

**EXTERNAL DEVICE**
Enter the source of the Current. Voltage excited terminals or lumped ports defined on boundaries in other physics interfaces are natural candidates but do not appear as options in the Voltage list because those do not have an accurate built-in current
measurement variable. You must select the User defined option and enter your own current variable, for example, using a suitable coupling operator (see Model Couplings in the COMSOL Multiphysics User’s Guide). Note that you must manually couple back the voltage variable in the electrical circuit to the context of the current measurement. This applies also when coupling to a voltage excited terminal or lumped port. The name of this voltage variable follows the convention cirn.UvsIm_v, where cirn is the tag of the Electrical Circuit interface node and UvsIm is the tag of the External U vs. I node. The mentioned tags are typically displayed within curly braces {} in the model tree.

**External I-Terminal**

The External I-Terminal node (\(\text{45}\)) connects an arbitrary voltage-to-ground measurement, for example, a circuit terminal boundary from another physics interface, as a voltage-to-ground assignment to a node in the electrical circuit. The resulting circuit current from the node is typically coupled back as a prescribed current source in the context of the voltage measurement. This feature does not apply when coupling to inductive or electromagnetic wave propagation models as then voltage must be defined as a line integral between two points rather than a single point measurement of electric potential. For such couplings, use the External I vs. U feature instead.

**NODE CONNECTIONS**

Set the Node name for the connecting node for the voltage assignment.

**EXTERNAL TERMINAL**

Enter the source of the Voltage. If you have circuit- or current-excited terminals defined on boundaries in other physics interfaces, these display as options in the Voltage list. You can also select the User defined option and enter your own voltage variable, for example, using a suitable coupling operator (see Model Couplings in the COMSOL Multiphysics User’s Guide). Note that, except for when coupling to a circuit terminal, you must manually couple back the current flow variable in the electrical circuit to the context of the voltage measurement. This applies also when coupling to a current excited terminal. The name of this current variable follows the convention cirn.termIm_i, where cirn is the tag of the Electrical Circuit interface node and termIm is the tag of the External I-Terminal node. The mentioned tags are typically displayed within curly braces {} in the model tree.
SPICE Circuit Import

By right-clicking the Electrical Circuit node you can import an existing SPICE netlist by selecting Import Spice Netlist. A window opens where you can enter a file location or browse your directories to find one. The default file extension for a SPICE netlist is .cir. The SPICE circuit import translates the imported netlist into Electrical Circuit interface nodes so these define the subset of SPICE features that can be imported.
Theory for the Electrical Circuit Interface

In this section:

- Electric Circuit Modeling and the Semiconductor Device Models
- NPN Bipolar Transistor
- n-Channel MOS Transistor
- Diode
- References for the Electrical Circuit Interface

Electric Circuit Modeling and the Semiconductor Device Models

Electrical circuit modeling capabilities are useful when simulating all sorts of electrical and electromechanical devices ranging from heaters and motors to advanced plasma reactors in the semiconductor industry. There are two fundamental ways that an electrical circuit model relates to a physical field model. Either the field model is used to get a better, more accurate description of a single device in the electrical circuit model or the electrical circuit is used to drive or terminate the device in the field model in such a way that it makes more sense to simulate both as a tightly coupled system.

The Electrical Circuit interface makes it possible to add nodes representing circuit elements directly to the model tree in a COMSOL Multiphysics model. The circuit variables can then be connected to a physical device model to perform co-simulations of circuits and multiphysics. The model acts as a device connected to the circuit so that you can analyze its behavior 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.

The circuit definition in COMSOL Multiphysics adheres to the SPICE format developed at University of California, Berkeley (Ref. 1) and SPICE netlists can also be imported, generating the corresponding features in the COMSOL Multiphysics model. Most circuit simulators can export to this format or some dialect of it.

There are three more advanced large-signal semiconductor device features available in the Electrical Circuit interface. Below, the equivalent circuits and the equations
defining their non-ideal circuit elements are given. For a more detailed account on semiconductor device modeling, see Ref. 2.

SEE ALSO

- NPN Bipolar Transistor
- n-Channel MOS Transistor
- Diode
- References for the Electrical Circuit Interface

**NPN Bipolar Transistor**

Figure 7-1 illustrates the equivalent circuit for the bipolar transistor.
Figure 7-1: A circuit for the bipolar transistor.

The following equations are used to compute the relations between currents and voltages in the circuit.
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.

\[
v_{rb} = \frac{1}{A} \left( R_{BM} - \frac{R_B - R_{BM}}{f_{bq}} \right) i_b
\]

\[
f_{bq} = \frac{1}{2 \left( 1 - \frac{u_{bc}}{V_{AF}} \right) \left( \frac{u_{be}}{V_{AR}} \right)} \left( 1 + \left[ \frac{1 + 4I_S e^{\frac{u_{tb}}{N_{b}V_T}} \left( \frac{u_{tb}}{N_{b}V_T} - 1 \right) + I_S e^{\frac{u_{tb}}{N_{e}V_T}} \left( \frac{u_{tb}}{N_{e}V_T} - 1 \right)}{I_{KF} A + I_{KRA} A} \right] \right)
\]

\[
\begin{align*}
   i_{be} &= A \left( \frac{I_S}{B_F} e^{-\frac{u_{ib}}{N_{b}V_T}} - 1 \right) + I_{SE} e^{-\frac{u_{ib}}{N_{e}V_T}} \left( \frac{u_{ib}}{N_{e}V_T} - 1 \right) \\
   i_{bc} &= A \left( \frac{I_S}{B_R} e^{-\frac{u_{ic}}{N_{b}V_T}} - 1 \right) + I_{SC} e^{-\frac{u_{ic}}{N_{e}V_T}} \left( \frac{u_{ic}}{N_{e}V_T} - 1 \right) \\
   i_{ce} &= A \left( \frac{I_S}{f_{bq}} e^{-\frac{u_{ie}}{N_{b}V_T}} + e^{-\frac{u_{ie}}{N_{e}V_T}} \right)
\end{align*}
\]

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.

\[
C_{jbx} = AC_{Jx} \times \left\{ \begin{array}{c}
\left( 1 - \frac{u_{bx}}{V_{Jx}} \right) \left[ -M_{Jx} \right] \\
\left( 1 - F_C \right)^{-1} \left[ 1 - M_{Jx} \right] \left( 1 - F_C (1 + M_{Jx}) + M_{Jx} \frac{u_{bx}}{V_{Jx}} \right)
\end{array} \right\}
\]

\[
\begin{align*}
   v_{bx} < F_C V_{Jx} \\
   v_{bx} \geq F_C V_{Jx}
\end{align*}
\]

The model parameters are listed in the table below.

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFAULT</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>( B_F )</td>
<td>100</td>
<td>Ideal forward current gain</td>
</tr>
<tr>
<td>( B_R )</td>
<td>1</td>
<td>Ideal reverse current gain</td>
</tr>
<tr>
<td>( C_{JC} )</td>
<td>0 F/m²</td>
<td>Base-collector zero-bias depletion capacitance</td>
</tr>
<tr>
<td>( C_{JE} )</td>
<td>0 F/m²</td>
<td>Base-emitter zero-bias depletion capacitance</td>
</tr>
<tr>
<td>( F_C )</td>
<td>0.5</td>
<td>Breakdown current</td>
</tr>
<tr>
<td>( I_{KF} )</td>
<td>Inf (A/m²)</td>
<td>Corner for forward high-current roll-off</td>
</tr>
<tr>
<td>( I_{KR} )</td>
<td>Inf (A/m²)</td>
<td>Corner for reverse high-current roll-off</td>
</tr>
<tr>
<td>( I_S )</td>
<td>1e-15 A/m²</td>
<td>Saturation current</td>
</tr>
<tr>
<td>( I_{SC} )</td>
<td>0 A/m²</td>
<td>Base-collector leakage saturation current</td>
</tr>
</tbody>
</table>
Table 7-1: Bipolar Transistor Model Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$I_{SE}$</td>
<td>0 A/m²</td>
<td>Base-emitter leakage saturation current</td>
</tr>
<tr>
<td>$M_{JC}$</td>
<td>1/3</td>
<td>Base-collector grading coefficient</td>
</tr>
<tr>
<td>$M_{JE}$</td>
<td>1/3</td>
<td>Base-emitter grading coefficient</td>
</tr>
<tr>
<td>$N_C$</td>
<td>2</td>
<td>Base-collector ideality factor</td>
</tr>
<tr>
<td>$N_E$</td>
<td>1.4</td>
<td>Base-emitter ideality factor</td>
</tr>
<tr>
<td>$N_F$</td>
<td>1</td>
<td>Forward ideality factor</td>
</tr>
<tr>
<td>$N_R$</td>
<td>1</td>
<td>Reverse ideality factor</td>
</tr>
<tr>
<td>$R_B$</td>
<td>0 Ωm²</td>
<td>Base resistance</td>
</tr>
<tr>
<td>$R_{BM}$</td>
<td>0 Ωm²</td>
<td>Minimum base resistance</td>
</tr>
<tr>
<td>$R_C$</td>
<td>0 Ωm²</td>
<td>Collector resistance</td>
</tr>
<tr>
<td>$R_E$</td>
<td>0 Ωm²</td>
<td>Emitter resistance</td>
</tr>
<tr>
<td>$T_{NOM}$</td>
<td>298.15 K</td>
<td>Device temperature</td>
</tr>
<tr>
<td>$V_{AF}$</td>
<td>Inf (V)</td>
<td>Forward Early voltage</td>
</tr>
<tr>
<td>$V_{AR}$</td>
<td>Inf (V)</td>
<td>Reverse Early voltage</td>
</tr>
<tr>
<td>$V_{JC}$</td>
<td>0.71 V</td>
<td>Base-collector built-in potential</td>
</tr>
<tr>
<td>$V_{JE}$</td>
<td>0.71 V</td>
<td>Base-emitter built-in potential</td>
</tr>
</tbody>
</table>

See Also

- Electric Circuit Modeling and the Semiconductor Device Models
- n-Channel MOS Transistor
- Diode
- References for the Electrical Circuit Interface

n-Channel MOS Transistor

Figure 7-2 illustrates an equivalent circuit for the MOS transistor.
Figure 7-2: A circuit for the MOS transistor.

The following equations are used to compute the relations between currents and voltages in the circuit.
There are also several capacitances between the terminals

\[ C_{bd} = C_{bd0} W \]
\[ C_{gs} = C_{gs0} W \]
\[ C_{jbd} = c_{BD} \times \begin{cases} 
(1 - F_C)^{-1} (1 - F_C (1 + M_J) + M_J \frac{V_{bd}}{F_B}) & v_{bd} < F_C P_B \\
(1 - M_J)^{-1} (1 + M_J + M_J \frac{V_{bd}}{F_B}) & v_{bd} \geq F_C P_B 
\end{cases} \]

The model parameters are as follows:

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFAULT</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>( c_{BD} )</td>
<td>0 F/m</td>
<td>Bulk-drain zero-bias capacitance</td>
</tr>
<tr>
<td>( c_{GDO} )</td>
<td>0 F/m</td>
<td>Gate-drain overlap capacitance</td>
</tr>
<tr>
<td>( c_{GSO} )</td>
<td>0 F/m</td>
<td>Gate-source overlap capacitance</td>
</tr>
<tr>
<td>( F_C )</td>
<td>0.5</td>
<td>Capacitance factor</td>
</tr>
<tr>
<td>( I_S )</td>
<td>1e-13 A</td>
<td>Bulk junction saturation current</td>
</tr>
<tr>
<td>( K_p )</td>
<td>2e-5 A/V^2</td>
<td>Transconductance parameter</td>
</tr>
<tr>
<td>( L )</td>
<td>50e-6 m</td>
<td>Gate length</td>
</tr>
<tr>
<td>( M_J )</td>
<td>0.5</td>
<td>Bulk junction grading coefficient</td>
</tr>
<tr>
<td>( N )</td>
<td>1</td>
<td>Bulk junction ideality factor</td>
</tr>
<tr>
<td>( P_B )</td>
<td>0.75 V</td>
<td>Bulk junction potential</td>
</tr>
</tbody>
</table>
**SEE ALSO**

- Electric Circuit Modeling and the Semiconductor Device Models
- NPN Bipolar Transistor
- Diode
- References for the Electrical Circuit Interface

**Diode**

Figure 7-3 illustrates equivalent circuit for the diode.
The following equations are used to compute the relations between currents and voltages in the circuit.

\[ i_{dh} = f(V_d) \]

\[ i_{rec} \]

\[ i_{dhl} \]

\[ V_d \]

\[ R_s \]

\[ C_j \]
\[ i_d = i_{dhl} + i_{drec} + i_{db} + i_e \]
\[ i_{dhl} = I_S \left( e^{\frac{v_d}{NVT}} - 1 \right) \left( 1 - \frac{1}{1 + I_S \left( \frac{v_d}{NVT} - 1 \right)} \right) \]
\[ i_{drec} = I_{SR} \left( e^{\frac{v_d}{NVT}} - 1 \right) \]
\[ i_{db} = I_{BV} e^{v_d + B_V} \]
\[ C_j = C_{J0} \times \begin{cases} \left( 1 - \frac{v_d}{V_J} \right)^M & v_d < F_C V_J \\ (1-M)^{-1} \left( 1 - F_C(1+M) + M \frac{v_d}{V_J} \right)^{1-M} & v_d \geq F_C V_J \end{cases} \]

where the following model parameters are required

### TABLE 7-3: DIODE TRANSISTOR MODEL PARAMETERS

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFAULT</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>( B_V )</td>
<td>Inf (V)</td>
<td>Reverse breakdown voltage</td>
</tr>
<tr>
<td>( C_{J0} )</td>
<td>0 F</td>
<td>Zero-bias junction capacitance</td>
</tr>
<tr>
<td>( F_C )</td>
<td>0.5</td>
<td>Forward-bias capacitance coefficient</td>
</tr>
<tr>
<td>( I_{BV} )</td>
<td>1e-09 A</td>
<td>Current at breakdown voltage</td>
</tr>
<tr>
<td>( I_{KP} )</td>
<td>Inf (A)</td>
<td>Corner for high-current roll-off</td>
</tr>
<tr>
<td>( I_S )</td>
<td>1e-13 A</td>
<td>Saturation current</td>
</tr>
<tr>
<td>( M )</td>
<td>0.5</td>
<td>Grading coefficient</td>
</tr>
<tr>
<td>( N )</td>
<td>1</td>
<td>Ideality factor</td>
</tr>
<tr>
<td>( N_{BV} )</td>
<td>1</td>
<td>Breakdown ideality factor</td>
</tr>
<tr>
<td>( N_R )</td>
<td>2</td>
<td>Recombination ideality factor</td>
</tr>
<tr>
<td>( R_S )</td>
<td>0 ( \Omega )</td>
<td>Series resistance</td>
</tr>
<tr>
<td>( T_{NOM} )</td>
<td>298.15 K</td>
<td>Device temperature</td>
</tr>
<tr>
<td>( V_J )</td>
<td>1.0 V</td>
<td>Junction potential</td>
</tr>
</tbody>
</table>

**SEE ALSO**
- Electric Circuit Modeling and the Semiconductor Device Models
• NPN Bipolar Transistor
• n-Channel MOS Transistor
• References for the Electrical Circuit Interface

References for the Electrical Circuit Interface

1. http://bwrc.eecs.berkeley.edu/Classes/IcBook/SPICE/

The AC/DC Module license includes an interface found under the Heat Transfer>Electromagnetic Heating branch in the Model Wizard. This interface combines magnetic fields with heat transfer.

In this chapter:
- The Induction Heating Interface
The Induction Heating Interface

The Induction Heating interface ( ), found under the Heat Transfer>Electromagnetic Heating branch ( ) in the Model Wizard, combines all features from the Magnetic Fields interface in the time-harmonic formulation with the Heat Transfer interface for modeling of induction and eddy current heating. The interface has the equations, boundary conditions, and sources for modeling. The predefined interaction adds the electromagnetic losses from the magnetic field as a heat source. This interface is based on the assumption that the magnetic cycle time is short compared to the thermal time scale (adiabatic assumption). Thus, it is associated with two predefined study types:

- **Frequency-Stationary**—time-harmonic magnetic fields and stationary heat transfer
- **Frequency-Transient**—time-harmonic magnetic fields and transient heat transfer

For more information about the underlying theory for this interface, see the Theory of Magnetic and Electric Fields in this guide and the Theory for the Heat Transfer Interfaces in the COMSOL Multiphysics User’s Guide.

When you add this interface, these default nodes are also added to the Model Builder—Induction Heating Model, Electromagnetic Heat Source, Boundary Electromagnetic Heat Source, Thermal Insulation, Magnetic Insulation, and Initial Values.

Right-click the Induction Heating node to add other features that implement boundary conditions and sources.

To display additional features for the physics interfaces and feature nodes, click the Show button ( ) in the Model Builder and select the applicable section.

*SHOW MORE OPTIONS FOR PHYSICS INTERFACES AND FEATURE NODES*

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

You can also click the Expand Sections button ( ) in the Model Builder to always show some sections or click the Show button ( ) and select Reset to Default to reset to display only the Equation and Override and Contribution sections.
For most physics feature nodes, both the **Equation** and **Override and Contribution** sections are always available. Click the **Show** button (++) and then select **Equation View** to display the **Equation View** node under all physics interface nodes in the **Model Builder**.

Availability of each feature, and whether it is described for a particular interface or node, is based on the individual physics interface and feature node. For example, the **Discretization**, **Advanced Settings**, **Consistent Stabilization**, and **Inconsistent Stabilization** sections are often described individually throughout the documentation as there are unique settings. See *Showing and Expanding Advanced Feature Nodes and Sections* in the **COMSOL Multiphysics User’s Guide** for additional links to the relevant documentation.

**INTERFACE IDENTIFIER**

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

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

**DOMAIN SELECTION**

Select the domains where you want to define the induction heating. The default setting is to include all domains in the model.

**DEPENDENT VARIABLES**

The dependent variables (field variables) are for the **Temperature** T and the **Magnetic Vector potential** A. You can change the name in the corresponding edit field, but the names of fields and dependent variables must be unique within a model.

**DISCRETIZATION**

To display this section, click the **Show** button (++) and select **Discretization**. Select **Quadratic**, **Linear**, **Cubic**, or **Quartic** for the **Temperature**, **Surface radiosity**, and **Magnetic vector potential**.

*Shared Feature Nodes for the Induction Heating Interface*

The **Induction Heating** interface shares most of its **Settings** windows with the **Magnetic Fields** (see *The Magnetic Fields Interface*) and the **Heat Transfer** interfaces.
The Heat Transfer in Solids, Heat Transfer in Fluids (general convection and conduction, non-isothermal flow, and conjugate heat transfer), and Joule Heating interfaces all belong to the COMSOL Multiphysics base package. See The Heat Transfer Interfaces and The Joule Heating Interface in the COMSOL Multiphysics User’s Guide for more information.

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

**Induction Heating Model**

The Induction Heating Model feature has settings to define the Conduction Current, Electric Field, Magnetic Field, Heat Conduction, and Thermodynamics.

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

**SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (shows) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**MODEL INPUTS**

This section contains field variables that appear as model inputs, if the current settings include such model inputs. By default, this section is empty. If you add a linear temperature relation for the conductivity, you can then define the source for the temperature $T$. From the Temperature list, select an existing temperature variable (from another physics interface) if available, or select User defined to define a value or expression for the temperature (SI unit: K) in the edit field that appears underneath the list.
CONDUCTION CURRENT
See the settings for Conduction Current under Ampère’s Law for the Magnetic Fields interface.

ELECTRIC FIELD
See the settings for Electric Field under Charge Conservation described for the Electrostatics interface.

MAGNETIC FIELD
See the settings for Magnetic Field under Ampère’s Law for the Magnetic Fields interface. Note the following difference:

- HB curve This option is not relevant for time harmonic modeling so it should not be selected when using the Induction Heating interface.

HEAT CONDUCTION
The default Thermal conductivity k (SI unit: W/(m·K)) uses values From material. If User defined is selected, choose Isotropic, Diagonal, Symmetric, or Anisotropic based on the characteristics of the thermal conductivity and enter other values or expressions in the field or matrix. The thermal conductivity k describes the relationship between the heat flux vector $\mathbf{q}$ and the temperature gradient $\nabla T$ as in $\mathbf{q} = -k\nabla T$, which is Fourier’s law of heat conduction. Enter this quantity as power per length and temperature.

THERMODYNAMICS
The default uses the Heat capacity at constant pressure $C_p$ (SI unit: J/(kg·K)) and Density $\rho$ (SI unit: kg/m$^3$) values From material. Select User defined to enter other values or expressions for one or both variables.

Electromagnetic Heat Source
The Electromagnetic Heat Source feature maps the electromagnetic losses as a heat source in the heat transfer part of the model. It appears as a default feature.

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

SHOW OR HIDE OPTIONS FOR PHYSICS FEATURE NODES
For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click
the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\(\text{Show}\)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Domain Selection**

Select the domains where you want to apply the model. The default feature setting is hard-coded to include all domains in the model.

**Initial Values**

The Initial Values feature adds initial values for the temperature, surface radiosity and magnetic vector potential.

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

**Show or Hide Options for Physics Feature Nodes**

For most physics interface feature nodes, the Equation and Override and Contribution sections are displayed on a feature node Settings window by default. You can also click the Expand Sections button on the Model Builder to always show some sections in an expanded view, or go to these menus to hide options as required. Click the Show button (\(\text{Show}\)) on the Model Builder and then select Equation View to display the Equation View node under all physics interface nodes in the Model Builder.

See the description for each physics interface for more links or go to Showing and Expanding Advanced Feature Nodes and Sections for more information.

**Domain Selection**

Select the domains where you want to apply the initial values. The default setting is to include all domains in the model.

**Initial Values**

Enter values or expressions for the Temperature \(T\) (SI unit: K), Surface radiosity \(J\) (SI unit: W/m\(^2\)) and Magnetic vector potential \(A\) (SI unit: Wb/m). The default temperature is 293.15 K.
This chapter describes the materials databases included with the AC/DC Module.

- Material Library and Databases
- Using the AC/DC Material Database
Material Library and Databases

**Note:** For detailed information about all the other materials databases and the separately purchased Material Library, see Materials in the *COMSOL Multiphysics User's Guide.*

In this section:

- About the Material Databases
- About Using Materials in COMSOL
- Opening the Material Browser
- Using Material Properties

*About the Material Databases*

All COMSOL modules have predefined material data available to build models. The most extensive material data is contained in the separately purchased Material Library, but all modules contain commonly used or module-specific materials. For example, the *Built-In* database is available to all users but the *MEMS* database is included with the...
MEMS Module and Structural Mechanics Module. You can also create your own materials and material libraries by researching and entering material properties yourself.

You access all the material databases (including the Material Library) from the Material Browser. These databases are briefly described below.

**RECENT MATERIALS**
From the Recent Materials folder ( ), you can select from a list of recently used materials, with the most recent at the top. This folder is available after the first time you add a material to a model.

**MATERIAL LIBRARY**
An optional add-on database, the Material Library ( ), contains data for over 2500 materials and 20,000 property functions.

**BUILT-IN**
Included with COMSOL Multiphysics, the Built-In database ( ) contains common solid materials with electrical, structural, and thermal properties. See Predefined Built-In Materials for all COMSOL Modules in the COMSOL Multiphysics User’s Guide for a list.

**AC/DC**
Included in the AC/DC Module, the AC/DC database ( ) has electric properties for some magnetic and conductive materials.

**BATTERIES AND FUEL CELLS**
Included in the Batteries & Fuel Cells Module, the Batteries and Fuel Cells database includes properties for electrolytes and electrode reactions for certain battery chemistries.

**LIQUIDS AND GASES**
Included in the Acoustics Module, Chemical Reaction Engineering Module, Subsurface Flow Module, Heat Transfer Module, and MEMS Module, the Liquids and Gases database ( ) includes transport properties and surface tension data for liquid/gas and liquid/liquid interfaces.
MEMS
Included in the MEMS Module and Structural Mechanics Module, the MEMS database has properties for MEMS materials—metals, semiconductors, insulators, and polymers.

PIEZOELECTRIC
Included in the Acoustics Module, MEMS Module, and Structural Mechanics Module, the Piezoelectric database has properties for piezoelectric materials.

USER-DEFINED LIBRARY
The User-Defined Library folder is where you create user-defined materials databases (libraries). When you create any new database, this also displays in the Material Browser. See Creating Your Own User-Defined Libraries in the COMSOL Multiphysics User’s Guide.

Note: All materials databases (including the Material Library) shipped with COMSOL Multiphysics and the optional modules are read-only.

About Using Materials in COMSOL

USING THE MATERIALS IN THE PHYSICS SETTINGS
The physics set-up in a model is determined by a combination of settings in the Materials and physics interface pages. When you add the first material to a model, COMSOL automatically assigns that material to the entire geometry. You can also select different geometric entities to have different materials. The following example
uses the *heat_sink.mph* model file contained in the Heat Transfer Module Model Library.

![Figure 9-1: Assigning materials to a heat sink model. Air is assigned as the material to the box surrounding the heat sink, and aluminum to the heat sink itself.](image)

If a geometry consists of a heat sink in a container, you can assign *Air* as the material in the container surrounding the heat sink and *Aluminum* as the heat sink material itself (see Figure 9-1). The *Conjugate Heat Transfer* physics interface, selected during model set-up, has a *Fluid* flow model, defined in the box surrounding the heat sink, and a *Heat Transfer* model, defined in both the aluminum heat sink and in the air box. The *Heat Transfer in Solids* 1 settings use the material properties associated to the *Aluminum 3003-H18* materials node, and the *Fluid* 1 settings define the flow using the *Air* material properties. The other nodes under *Conjugate Heat Transfer* define the initial and boundary conditions.

All physics interface properties automatically use the correct *Materials* properties when you use the default *From material* setting. This means that you can use one node to define the physics across several domains with different materials; COMSOL then uses the material properties from the different materials to define the physics in the domains. If material properties are missing, the *Material Contents* section on the *Materials* page displays a stop icon (■) to warn you about the missing properties and a warning icon (⚠️) if the property exists but its value is undefined. See The Material Page in the COMSOL Multiphysics User’s Guide.

There are also some physics interface properties that by default define a material as the *Domain material* (that is, the materials defined on the same domains as the physics
interface). For such material properties you can also select any other material that is present in the model, regardless of its selection.

**Evaluating and Plotting Material Properties**

You can access the material properties like other variables in a model using the variable naming conventions and scoping mechanisms. To access a material property in a plot or evaluation, you need to know the tags for the material and the property group. Typically, for the first material (Material 1) the tag is `mat1` and most properties reside in the default Basic property group with the tag `def`. The variable names appear in the **Variable** column in the table under **Output properties** in the **Settings** window for the property group; for example, `Cp` for the heat capacity at constant pressure. The syntax for referencing the heat capacity at constant pressure in Material 1 is then `mat1.def.Cp`. Some properties are anisotropic tensors, and you can then access each of the components, such as `mat1.def.k11`, `mat1.def.k12`, and so on, for the thermal conductivity. For material properties that are functions, you need to call them with input arguments such as `mat1.def.rho(pA,T)` where `pA` and `T` are numerical values or variables representing the absolute pressure and the temperature, respectively. You can also plot the functions directly from the function nodes’ **Settings** window by first specifying suitable ranges for the input arguments.

Many physics interfaces also define variables for the material properties that they use. For example, `solid.rho` is the density in the Solid Mechanics interface and is equal to the density in a material where it is used in the domains where the Solid Mechanics interface is active.

**Opening the Material Browser**

---

**Note**: When you are using the **Material Browser**, the words **window** and **page** are interchangeable. For simplicity, the instructions refer only to the **Material Browser**.

1. Open or create a model file.
2. From the **Options** menu select **Material Browser** or right-click the **Materials** node and select **Open Material Browser**.

   The **Material Browser** opens by default to the right of the **Settings** window.
3 Under **Material Selection**, search or browse for materials.

- Enter a **Search** term to find a specific material by name, UNS number (Material Library materials only), or DIN number (Material Library materials only). If the search is successful, a list of filtered databases containing that material displays under **Material Selection**.

**Note:** To clear the search field and browse, delete the search term and click **Search** to reload all the databases.

- Click to open each database and browse for a specific material by class (for example, in the Material Library) or physics module (for example, MEMS Materials).

**Note:** Always review the material properties to confirm they are applicable for the model. For example, **Air** provides temperature-dependent properties that are valid at pressures around 1 atm.

4 When you locate the material you want, right-click to **Add Material to Model**.

A node with the material name is added to the **Model Builder** and the **Material** page opens.

**Using Material Properties**

For detailed instructions, see **Adding Predefined Materials** and **Material Properties Reference** in the *COMSOL Multiphysics User’s Guide*. 
Using the AC/DC Material Database

All physics interfaces in the AC/DC Module support the use of the COMSOL Multiphysics material databases. The electromagnetic material properties that you can store in the material databases are:

- Electrical conductivity and resistivity
- Relative permittivity
- Relative permeability
- Nonlinear BH-curves
- Refractive index

The AC/DC database is included with the AC/DC Module and contains electromagnetic and other material properties for these materials:

**PREDEFINED MATERIALS**
- Copper
- Soft Iron (without losses)
- Soft Iron (with losses)
- Quartz
- Graphite
- Graphite felt
- Silicon Carbide

Some properties depend on the magnetic flux density, location, or temperature. The database contains, depending on the material and in addition to the more common material properties, the following properties:

**PREDEFINED PROPERTIES**
- Remnant flux density
- Reference temperature
- Temperature coefficient
- Nonlinear BH-curves
- Resistivity at reference temperature
This Glossary of Terms contains finite element modeling terms in an electromagnetics context. For mathematical terms as well as geometry and CAD terms specific to the COMSOL Multiphysics software and documentation, please see the glossary in the COMSOL Multiphysics User’s Guide. For references to more information about a term, see the index.
Glossary of Terms

**anisotropy**  Variation of material properties with direction.

**constitutive relation**  The relation between the $D$ 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**  Two equal and opposite charges $+q$ and $-q$ separated a short distance $d$. The electric dipole moment is given by $p = qd$, where $d$ 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 $m = IAe$, where $I$ is the current carried by the loop, $A$ its area, and $e$ a unit vector along the central axis of the loop.

**Nedelec's edge element**  See vector element.

**phasor**  A complex function of space representing a sinusoidally varying quantity.

**quasi-static approximation**  The electromagnetic fields are assumed to vary slowly, so that the retardation effects can be neglected. This approximation is valid when the geometry under study is considerably smaller than the wavelength.

**vector element**  A finite element often used for electromagnetic vector fields. The tangential component of the vector field at the mesh edges is used as a degree of freedom. Also called Nedelec's edge element or just edge element.
Index

2D axisymmetric models
- coil domains 176
- guidelines for solving 60
- multi-turn coils 172
- single-turn coils 174
- transient equation 223

2D models
- coil domains 176
- guidelines for solving 60
- multi-turn coils 172
- single-turn coils 174
- transient equation 223

3D models
- guidelines for solving 61
- importing GDS-II files 87
- 3D models, infinite elements and 70

A
- AC/DC physics interfaces 14
- adiabatic assumption 64
- admittance, coils and 180
- air stress tensors 43, 47
- Ampère’s law (node) 166
- Ampère’s law and current conservation (node) 229
- anisotropic materials 34–35
- applying, electromagnetic sources 63
- Archie’s law (node) 127
- axial symmetry, cylindrical coordinates 61

B
- bond wires 89–91
- boundary conditions
  - electric currents interface 131
  - electric currents, shell interface 148
  - electromagnetics theory 36
  - electrostatics interface 106
  - forced voltage, port 75
- capacitive network 75–76
- capacitance matrix 75–76
- capacitor (node) 240
- Cartesian coordinates 18, 60, 68
- cementation exponent 128, 161
- charge conservation (node) 100
- charge relaxation theory 151–153
- circuit connections, coils 174–175, 178
- circuit import, SPICE 248
- coil domain, multi-turn 172
- coil domain, single-turn 174
- coil group domain (node) 176
- conductive media 147, 162
- constitutive relations theory 27, 29
- contact impedance (node) 140
- contacting COMSOL 20
- continuity (node)
  - electric currents interface 143
  - electrostatics interface 118
  - magnetic fields interface 195
- continuum mechanics forces 39
- coordinate systems, infinite elements and 51
- current sources 73
- current sources import 52
- current sources, passives 52
- current sources, passive 63
- current conservation (node) 100
- current conservation interface 132
- current source, node 127
- current source, boundary 132
- current source, built-in 127
- current source, node 127
- current sources, passives 52
- current sources, passive 63
- circuit connections, coils 174–175, 178
- circuit import, SPICE 248
- coil domain, multi-turn 172
- coil domain, single-turn 174
- coil group domain (node) 176
- conductive media 147, 162
- constitutive relations theory 27, 29
- contact impedance (node) 140
- contacting COMSOL 20
- continuity (node)
  - electric currents interface 143
  - electrostatics interface 118
  - magnetic fields interface 195
- continuum mechanics forces 39
- coordinate systems, infinite elements and 51

C
- calculating
  - electromagnetic forces 39, 73
  - forces in moving objects 46
  - lumped parameters 52
  - mapped infinite elements 51
  - calculating S-parameters 82
- capacitance matrix 75–76
- capacitor (node) 240
- Cartesian coordinates 18, 60, 68
- cementation exponent 128, 161
- charge conservation (node) 100
- charge relaxation theory 151–153
- circuit connections, coils 174–175, 178
- circuit import, SPICE 248
- coil domain, multi-turn 172
- coil domain, single-turn 174
- coil group domain (node) 176
- conductive media 147, 162
- constitutive relations theory 27, 29
- contact impedance (node) 140
- contacting COMSOL 20
- continuity (node)
  - electric currents interface 143
  - electrostatics interface 118
  - magnetic fields interface 195
- continuum mechanics forces 39
- coordinate systems, infinite elements and 51
<table>
<thead>
<tr>
<th>Index Term</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coulomb gauge</td>
<td>217–218</td>
</tr>
<tr>
<td>current conservation (node)</td>
<td>124</td>
</tr>
<tr>
<td>current conservation, ungauged formulations</td>
<td>220</td>
</tr>
<tr>
<td>current source (node)</td>
<td></td>
</tr>
<tr>
<td>boundary, electric currents interface</td>
<td>132</td>
</tr>
<tr>
<td>electric current interface</td>
<td>129</td>
</tr>
<tr>
<td>electrical circuit interface</td>
<td>241</td>
</tr>
<tr>
<td>current sources</td>
<td></td>
</tr>
<tr>
<td>line 144</td>
<td>153</td>
</tr>
<tr>
<td>current-controlled current source (node)</td>
<td>243</td>
</tr>
<tr>
<td>current-controlled voltage source (node)</td>
<td>242</td>
</tr>
<tr>
<td>current-driven coils</td>
<td>173, 175, 177</td>
</tr>
<tr>
<td>cylindrical coordinates</td>
<td>18, 60</td>
</tr>
<tr>
<td>D device models, electrical circuits</td>
<td>249</td>
</tr>
<tr>
<td>dielectric shielding (node)</td>
<td>111</td>
</tr>
<tr>
<td>dielectrics and perfect conductors</td>
<td>36</td>
</tr>
<tr>
<td>diode (node)</td>
<td>245</td>
</tr>
<tr>
<td>diode transistor model</td>
<td>256</td>
</tr>
<tr>
<td>dispersive materials</td>
<td>35</td>
</tr>
<tr>
<td>displacement field (node)</td>
<td>114</td>
</tr>
<tr>
<td>distributed capacitance (node)</td>
<td>115</td>
</tr>
<tr>
<td>distributed impedance (node)</td>
<td>135</td>
</tr>
<tr>
<td>documentation, location</td>
<td>19</td>
</tr>
<tr>
<td>domain material</td>
<td>271</td>
</tr>
<tr>
<td>drill files</td>
<td>89</td>
</tr>
<tr>
<td>E ECAD import</td>
<td>84</td>
</tr>
<tr>
<td>options</td>
<td>91</td>
</tr>
<tr>
<td>troubleshooting</td>
<td>94</td>
</tr>
<tr>
<td>edge current (node)</td>
<td>234</td>
</tr>
<tr>
<td>effective conductivity, porous media</td>
<td>157</td>
</tr>
<tr>
<td>effective relative permittivity</td>
<td>159</td>
</tr>
<tr>
<td>elastic material stress tensor</td>
<td>45, 48</td>
</tr>
<tr>
<td>elastic pure conductor</td>
<td>45, 48</td>
</tr>
<tr>
<td>electric currents interface</td>
<td>122</td>
</tr>
<tr>
<td>theory</td>
<td>156</td>
</tr>
<tr>
<td>electric currents, shell interface</td>
<td>146</td>
</tr>
<tr>
<td>theory</td>
<td>162</td>
</tr>
<tr>
<td>electric field transformation (node)</td>
<td>212</td>
</tr>
<tr>
<td>electric fields theory</td>
<td>151</td>
</tr>
<tr>
<td>electric forces and torques</td>
<td>50</td>
</tr>
<tr>
<td>electric insulation (node)</td>
<td>139</td>
</tr>
<tr>
<td>electric point dipole (node)</td>
<td>145, 197</td>
</tr>
<tr>
<td>electric potential (node)</td>
<td>109</td>
</tr>
<tr>
<td>electric shielding (node)</td>
<td>136</td>
</tr>
<tr>
<td>electrical circuit interface</td>
<td>238</td>
</tr>
<tr>
<td>theory</td>
<td>249</td>
</tr>
<tr>
<td>electrical conductivity, porous media</td>
<td>160</td>
</tr>
<tr>
<td>electrical size</td>
<td>59</td>
</tr>
<tr>
<td>electromagnetic energy, theory</td>
<td>31</td>
</tr>
<tr>
<td>electromagnetic forces and torques</td>
<td>72</td>
</tr>
<tr>
<td>electromagnetic forces, calculating</td>
<td>39</td>
</tr>
<tr>
<td>electromagnetic heat source (node)</td>
<td>265</td>
</tr>
<tr>
<td>electromagnetic quantities</td>
<td>53</td>
</tr>
<tr>
<td>electromagnetic sources, applying</td>
<td>63</td>
</tr>
<tr>
<td>electromagnetic stress tensors</td>
<td>43</td>
</tr>
<tr>
<td>electrostatic point dipole (node)</td>
<td>120</td>
</tr>
<tr>
<td>electrostatics interface</td>
<td>98</td>
</tr>
<tr>
<td>theory</td>
<td>155</td>
</tr>
<tr>
<td>emailing COMSOL</td>
<td>20</td>
</tr>
<tr>
<td>external current density (node)</td>
<td></td>
</tr>
<tr>
<td>electric currents interface</td>
<td>128</td>
</tr>
<tr>
<td>Magnetic Fields interface</td>
<td>170</td>
</tr>
<tr>
<td>external I vs. U (node)</td>
<td>245</td>
</tr>
<tr>
<td>external I-terminal (node)</td>
<td>247</td>
</tr>
<tr>
<td>external U vs. I (node)</td>
<td>246</td>
</tr>
<tr>
<td>extruding layers</td>
<td>84</td>
</tr>
<tr>
<td>F field variables in 2D</td>
<td>64</td>
</tr>
<tr>
<td>file formats</td>
<td></td>
</tr>
<tr>
<td>GDS-II</td>
<td>86</td>
</tr>
<tr>
<td>NETEX-G</td>
<td>88</td>
</tr>
<tr>
<td>file, Touchstone</td>
<td>77</td>
</tr>
</tbody>
</table>
fixed current, ports 75
floating potential (node) 113
fluid saturation 128
force calculation (node) 102
forced voltage, port 75
forces
calculating 72
continuum mechanics and 39
elastic solids and 41
electromagnetic, calculating 39
in moving objects 46
stationary fields 43
torque 42
formation factor 161
frequency domain study 153, 158
G Galilei invariants and transformations 47
gauge fixing for A-field (node) 169
gauge transformation and fixing 217, 219
Gauss’ law and charge relaxation theory 151
Gauss’ law equation 155
GDS-II file format 86
general elastic materials 45, 48
geometry, simplifying 59
Gerber layer files 88
ground (node) 107
ground node (node) 240
H Helmholtz’s theorem 217
I impedance boundary condition (node) 188
impedance, coils and 179
importing
ECAD files 84
GDS-II files 86
NETEX-G files 88
OBD++(X) files 85
SPICE netlists 248
wirebonds 89
inductance, coils and 179
induction heating interface 262
induction heating model (node) 264
inductor (node) 240
infinite elements (node) 103
infinite elements, calculating 51
inhomogeneous materials 34
initial values (node)
electric currents interface 130
electrostatics interface 105
induction heating interface 266
magnetic and electric fields interface 230
magnetic fields interface 180
magnetic fields, no currents interface 202
rotating machinery, magnetic interface 212
Internet resources 18
K knowledge base, COMSOL 20
L layers, extruding 84
line charge (node) 119
line current (out of plane) (node) 196
line current source (node) 143
Lorentz forces, calculating 73
Lorentz term 33
lumped parameters
calculating 74–77
converting 52
lumped port (node) 185
lumped ports 79
M magnetic and electric fields interface 226
theory 235
magnetic field (node) 184
Magnetic Fields interface
theory 222
magnetic fields interface 164
magnetic fields, no currents interface 199
theory 224
magnetic flux conservation (node) 201
magnetic flux density (node) 204
magnetic forces and torques 49
magnetic insulation (node) 182, 206
magnetic point dipole (node) 198
magnetic potential (node) 187
magnetic shielding (node) 206
manual scaling (node) 104
mapped infinite elements 51, 67
Material Browser 272
Material Library 269
materials
databases 269
domain, default 271
grouping of 34
properties, evaluating and plotting 272
using 274
Maxwell stress tensor, calculating 72
Maxwell’s equations 36, 249
quasi-static approximation 215
theory 26
mesh resolution 65
method of virtual work 49
Model Library example
boundary conditions 62
cartesian coordinates 60
stress tensors 45
symmetries 61
total forces 45
Model Library examples
electromagnetic forces 73
Model Library, accessing in COMSOL 19
model M-file 14
MPH-files 19
multi-turn coil domain (node) 172

N
n-Channel MOS transistor 244, 253
n-Channel MOSFET (node) 244
NETEX-G file format 88
netlists, SPICE 248–249
nonlinear materials 35
normal current density (node) 133
NPN bipolar junction transistor 244, 250
NPN BJT (node) 244
O
ODB++(X) files 85
Ohm’s law and charge relaxation theory
151
opening the Model Library 19
P
parameters, infinite elements 69
PDE formulations 30
perfect conductors and dielectrics 36
perfect magnetic conductor (node) 190
periodic condition (node) 116
magnetic fields interface 194
permittivity, volume average 159
phasors 37
plotting
material properties 272
PMAC. see perfect magnetic conductor.
point charge (node) 120
point current source (node) 144
point dipoles
electric 197
magnetic 198
porous media and mixtures 123
port boundary conditions 82
port sweeps and lumped parameters 77
ports, lumped 79
potentials, scalar and magnetic 30
power law, porous media 158, 160
power, coils and 178
Poyning’s theorem 32
prescribed rotation (node) 213
prescribed rotational velocity (node) 214
<table>
<thead>
<tr>
<th>Index</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>principle of virtual displacement</td>
<td>49</td>
</tr>
<tr>
<td>pure conductor stress tensor</td>
<td>45, 48</td>
</tr>
<tr>
<td>quasi-static approximation</td>
<td>33, 215</td>
</tr>
<tr>
<td>reciprocal permittivity, volume average</td>
<td>160</td>
</tr>
<tr>
<td>relative repair tolerance</td>
<td>93</td>
</tr>
<tr>
<td>resistance, coils and</td>
<td>179</td>
</tr>
<tr>
<td>resistor (node)</td>
<td>240</td>
</tr>
<tr>
<td>reversed current direction (node)</td>
<td>180</td>
</tr>
<tr>
<td>rotating machinery, magnetic interface</td>
<td>210</td>
</tr>
<tr>
<td>saturation coefficient</td>
<td>161</td>
</tr>
<tr>
<td>saturation exponent</td>
<td>128</td>
</tr>
<tr>
<td>scattering parameters, see S-parameters</td>
<td></td>
</tr>
<tr>
<td>sector symmetry (node)</td>
<td></td>
</tr>
<tr>
<td>electric currents interface</td>
<td>142, 195</td>
</tr>
<tr>
<td>selecting</td>
<td></td>
</tr>
<tr>
<td>solvers</td>
<td>65</td>
</tr>
<tr>
<td>space dimensions</td>
<td>59</td>
</tr>
<tr>
<td>study types</td>
<td>63</td>
</tr>
<tr>
<td>semiconductor device models</td>
<td>249</td>
</tr>
<tr>
<td>SI units</td>
<td>53</td>
</tr>
<tr>
<td>single-turn coil domain (node)</td>
<td>174</td>
</tr>
<tr>
<td>skin effects</td>
<td>65</td>
</tr>
<tr>
<td>solver settings</td>
<td>65</td>
</tr>
<tr>
<td>space charge density (node)</td>
<td>102</td>
</tr>
<tr>
<td>space dimensions, selecting</td>
<td>59</td>
</tr>
<tr>
<td>S-parameter calculations</td>
<td>82</td>
</tr>
<tr>
<td>SPICE netlists</td>
<td>248–249</td>
</tr>
<tr>
<td>stationary fields, forces</td>
<td>43</td>
</tr>
<tr>
<td>stress tensors</td>
<td>43, 45, 48</td>
</tr>
<tr>
<td>study types</td>
<td></td>
</tr>
<tr>
<td>electric currents interface</td>
<td>158</td>
</tr>
<tr>
<td>electromagnetic theory</td>
<td>30</td>
</tr>
<tr>
<td>induction heating interface</td>
<td>262</td>
</tr>
<tr>
<td>physics interface availability</td>
<td>17</td>
</tr>
<tr>
<td>selecting</td>
<td>63</td>
</tr>
<tr>
<td>subcircuit definition (node)</td>
<td>243</td>
</tr>
<tr>
<td>subcircuit instance (node)</td>
<td>244</td>
</tr>
<tr>
<td>surface charge density (node)</td>
<td>110</td>
</tr>
<tr>
<td>surface current (node)</td>
<td>184</td>
</tr>
<tr>
<td>technical support, COMSOL</td>
<td>20</td>
</tr>
<tr>
<td>terminal (node)</td>
<td>112</td>
</tr>
<tr>
<td>constitutive relations</td>
<td>27, 29</td>
</tr>
<tr>
<td>electric currents</td>
<td>156</td>
</tr>
<tr>
<td>electric currents, shell interface</td>
<td>162</td>
</tr>
<tr>
<td>electric fields</td>
<td>151</td>
</tr>
<tr>
<td>electrical circuit</td>
<td>249</td>
</tr>
<tr>
<td>electromagnetics</td>
<td>26</td>
</tr>
<tr>
<td>electrostatics interface</td>
<td>155</td>
</tr>
<tr>
<td>lumped ports</td>
<td>79</td>
</tr>
<tr>
<td>magnetic and electric fields</td>
<td>215</td>
</tr>
<tr>
<td>magnetic and electric fields interface</td>
<td>235</td>
</tr>
<tr>
<td>magnetic fields, no currents interface</td>
<td>224</td>
</tr>
<tr>
<td>thin low permeability gap (node)</td>
<td></td>
</tr>
<tr>
<td>magnetic fields interface</td>
<td>193</td>
</tr>
<tr>
<td>magnetic fields, no currents interface</td>
<td>207</td>
</tr>
<tr>
<td>thin low permittivity gap (node)</td>
<td>117</td>
</tr>
<tr>
<td>thin shells, conductive media</td>
<td>162</td>
</tr>
<tr>
<td>time dependent study</td>
<td>63, 153, 158</td>
</tr>
<tr>
<td>tolerance, relative repair</td>
<td>93</td>
</tr>
<tr>
<td>torque forces</td>
<td>42</td>
</tr>
<tr>
<td>torques, calculating</td>
<td>72</td>
</tr>
<tr>
<td>Touchstone and port sweeps</td>
<td>77</td>
</tr>
<tr>
<td>transformations, Galilei</td>
<td>47</td>
</tr>
<tr>
<td>transient study</td>
<td>63</td>
</tr>
<tr>
<td>transition boundary condition (node)</td>
<td>191</td>
</tr>
<tr>
<td>typographical conventions</td>
<td>20</td>
</tr>
</tbody>
</table>
unbounded domains, modeling 67
units, SI 53
user community, COMSOL 20

vacuum stress tensors 43, 47
variables
S-parameters 83
variables, physical quantities 53
velocity (Lorentz term) (node) 171
voltage input, ports 79
voltage source (node) 241
voltage-controlled current source
  (node) 242
voltage-controlled voltage source (node)
  242
voltage-driven coils 173, 175, 177
volume averages, porous media 157

wavelength 65
web sites, COMSOL 20
wirebonds, importing 89–91

zero charge (node) 117
zero magnetic scalar potential (node)
  205