MEMS Module

User's Guide

COMSOL

MEMS Module User's Guide

Contact Information

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

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

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

Part number: CM020901

Contents

Chapter 1: Introduction

Chapter 2: Creating and Analyzing MEMS Models

Chapter 3: AC/DC Interfaces

Chapter 4: Fluid Flow Interfaces

Chapter 5: Structural Mechanics Interfaces

Chapter 6: Piezoresistivity Interfaces

Chapter 7: Glossary

1

Introduction

This user's guide describes the MEMS Module. The *MEMS Module User's Guide* introduces the modeling stages in COMSOL Multiphysics® and this module and serves as a reference for more advanced modeling techniques and details about the physics interfaces.

In this chapter:

- About the MEMS Module
- Overview of the User's Guide

About the MEMS Module

In this section:

- About MEMS
- About the MEMS Module
- The MEMS Module Physics Interface Guide
- Common Physics Interface and Feature Settings and Nodes
- Coupling to Other Physics Interfaces
- Where Do I Access the Documentation and Application Libraries?

	The Physics Interfaces and Building a COMSOL Multiphysics Model in the *COMSOL Multiphysics Reference Manual*

About MEMS

The field of MEMS evolved as engineers and scientists explored new avenues to make use of the fabrication technologies developed by the microelectronics industry. These technologies enabled complex micron and submicron structures to be integrated with electronic systems and batch fabricated at low cost. Mechanical devices fabricated using these technologies have become known as *microelectromechanical systems* (MEMS) or alternatively as microsystems. A proper description of these devices usually requires multiple physical effects to be incorporated.

At the microscale, different physical effects become important to those dominant at macroscopic scales. Inertial forces, which scale with the volume of the system, become comparatively less important than effects that scale more favorably when the system size is reduced, such as electrostatic forces. Consequently electrostatically actuated devices have been developed to measure acceleration and rotation with high accuracy in a small package. These accelerometers and gyroscopes have found widespread application in the automotive industry (where they are used to deploy airbags), in industrial applications (as sensors), and in consumer applications (where smartphones have driven rapid growth in the use of MEMS devices in recent years). Capacitive sensing has also led to widespread adoption of capacitive pressure sensors, which have largely superseded the piezoresistive pressure sensors that were some of the first MEMS devices to come to market. Piezoelectric MEMS devices have also become

common with the widespread adoption of FBAR (thin–film bulk acoustic resonator) technology in mobile communication applications. Although there are few commercially important devices that employ thermal actuation (largely due to the high power consumption of such devices), thermal phenomena, such as packaging-induced thermal stresses, are often important in the design of MEMS devices. Finally, for resonant MEMS devices such as accelerometers, damping phenomena are important and there is a need for detailed modeling of phenomena such as thin-film damping. Very high quality factor resonators are often used in MEMS timing applications, and for these devices the quality factor can be limited by thermoelasticity: the intrinsic coupling of the thermal and structural domains that exists as a result of the laws of thermodynamics.

Microfluidic systems are often included within the field of MEMS and represent a growing market for MEMS devices. For modeling microfluidics devices, the dedicated Microfluidics Module contains tailored physics interfaces for the simulation of microfluidic devices, including creeping flow, multiphase flow, electrokinetic effects, and slip flow.

About the MEMS Module

The MEMS Module is a collection of physics interfaces for the simulation of MEMS devices. It has a range of tools to address the specific challenges of modeling microsystems. It includes physics interfaces to model electromechanics (used for modeling electrostatically actuated structures), solid mechanics, piezoelectricity, piezoresistivity, and electrical circuits (which can be straightforwardly coupled to domain models). There are several predefined interfaces to model thermal-structural interactions, including the Thermoelasticity interface (for modeling damping in high quality factor resonators), Joule Heating with Thermal Expansion (for modeling thermal actuators), and the Thermal Stress interface (used for modeling thermal stresses when thermal gradients are present within devices). The Fluid-Structure Interaction interface implements a fully bidirectional coupling between fluid flow and structural motion, including the viscous, pressure, and inertial fluid forces as well as momentum transfer from the solid to the fluid. Finally the Thin-Film Flow interfaces allow thin layers of fluid to be modeled in an efficient manner and can be straightforwardly coupled to structural displacement to model damping phenomena, even under rarefied conditions.

COMSOL Multiphysics is uniquely transparent because the model equations are always visible and you have complete freedom in the definition of phenomena that are not predefined in the module.

The MEMS Module Applications Libraries and supporting documentation explain how to use the physics interfaces to model a range of MEMS devices.

	• Coupling to Other Physics Interfaces
	• Physics and Scaling for MEMS Devices
	• The MEMS Module Study Capabilities by Physics Interface

The MEMS Module Physics Interface Guide

The MEMS Module extends the functionality of the physics interfaces of the base package for COMSOL Multiphysics. The details of the physics interfaces and study types for the MEMS Module are listed in the table. The functionality of the COMSOL Multiphysics base package is given in the *COMSOL Multiphysics Reference Manual*.

In the *COMSOL Multiphysics Reference Manual*:

• Studies and Solvers

• The Physics Interfaces

• For a list of all the core physics interfaces included with a COMSOL Multiphysics license, see Physics Interface Guide.

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

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Electrical Circuit		cir	Not space dependent	stationary; frequency domain; time dependent; frequency domain; eigenfrequency
Electrostatics[1]		es	all dimensions	stationary; time dependent; stationary source sweep; eigenfrequency; frequency domain; small signal analysis, frequency domain
Elastic Waves				
Elastic Waves, Time Explicit		elte	3D, 2D, 2D axisymmetric	time dependent
Piezoelectric Waves, Time Explicit		—	3D, 2D, 2D axisymmetric	time dependent
Fluid Flow				
Single-Phase Flow				
Laminar Flow[1]		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Fluid-Structure Interaction				
Fluid-Solid Interaction[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Conjugate Heat Transfer, Fluid-Solid Interaction[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent;
Fluid-Solid Interaction, Viscoelastic Flow[2,7]		—	3D, 2D, 2D axisymmetric	time dependent; time dependent with phase initialization
Fluid-Solid Interaction, Two-Phase Flow, Phase Field[2, 6]		—	3D, 2D, 2D axisymmetric	time dependent; time dependent with phase initialization

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Thin-Film Flow				
Thin-Film Flow		tff	3D, 2D, 2D axisymmetric	stationary; time dependent; frequency domain; eigenfrequency
Thin-Film Flow, Domain		tff	2D	stationary; time dependent; frequency domain; eigenfrequency
Electromagnetic Heating				
Joule Heating[1,2]		—	all dimensions	stationary; time dependent; small-signal analysis; frequency domain
Pyroelectricity		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Structural Mechanics				
Solid Mechanics[1]		solid	3D, 2D, 2D axisymmetric, 1D, 1D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; mode analysis; time dependent; time dependent, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; frequency domain, modal reduced-order model; frequency domain, AWE reduced-order model; response spectrum; random vibration (PSD); linear buckling
Thermal Stress, Solid[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Joule Heating and Thermal Expansion[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Thermoelasticity		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; time dependent
Electromechanics		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency, prestressed; time dependent; frequency domain, prestressed
Electromechanics, Boundary Elements		—	3D, 2D	stationary; eigenfrequency, prestressed; time dependent; frequency domain, prestressed
Solid Thin-Film Damping		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; time dependent, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, modal reduced-order model; time dependent; response spectrum; random vibration (PSD)
Shell Thin-Film Damping		—	3D, 2D axisymmetric	stationary; eigenfrequency; time dependent; time dependent, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, modal reduced-order model; time dependent; response spectrum; random vibration (PSD)

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Piezoelectricity[2]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, modal; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; small-signal analysis, frequency domain; linear buckling
Piezoelectricity, Layered Shell[2 5]		—	3D	stationary; eigenfrequency; time dependent; frequency domain
Piezoelectricity and Pyroelectricity		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, modal; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; small-signal analysis, frequency domain; linear buckling
Magnetostriction[2,3]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Magnetomechanics[2,3]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed
Magnetomechanics, No Currents[2,3]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed
Ferroelectroelasticity[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent; frequency domain
Electrostriction[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent; frequency domain
Piezoresistivity				
Piezoresistivity, Domain Currents		—	3D	stationary; eigenfrequency; time dependent; time dependent, modal; frequency domain; frequency domain, modal; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed
Piezoresistivity, Shell[4]		—	3D	stationary; time dependent; time dependent, modal; frequency domain; frequency domain, modal; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Piezoresistivity, Boundary Currents		—	3D	stationary; eigenfrequency; time dependent; time dependent, modal; frequency domain; frequency domain, modal; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed

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

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

[3] Requires the addition of the AC/DC Module.

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

[5] Requires the addition of the Composite Materials Module.

[6] Requires the addition of the CFD Module, or the Polymer Flow Module or the Microfluidics Module.

[7] Requires the addition of the Polymer Flow Module.

Common Physics Interface and Feature Settings and Nodes

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

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

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

Coupling to Other Physics Interfaces

The MEMS Module connects to COMSOL Multiphysics and other add-on modules in the COMSOL Multiphysics product line. You can view and modify the models in terms of the underlying PDEs. Table 1-1 summarizes the most important MEMS couplings and other common devices you can model using this module.

The first column of the table lists phenomena, couplings, and devices that are often associated with the word electromechanical in the narrow and literal meaning of MEMS. The devices in this category are usually various kinds of actuators and sensors. The microfluidic devices, although using some of the same manufacturing and miniaturizing techniques, form a totally different application area. The *Fluid-Structure Interaction* column lists various techniques and phenomena that are useful for both electromechanical and microfluidic applications, where movement and deformation of solids are of concern.

This table, however, shows only the tip of the iceberg — our view of the most important applications where you can use the MEMS Module. In your hands, the multiphysics combinations and applications are unlimited.

TABLE 1-1: EXAMPLES OF COUPLED PHENOMENA AND DEVICES YOU CAN MODEL USING THE MEMS MODULE

	ELECTROMECHANICAL	FLUID-STRUCTURE INTERACTION
PHENOMENON/ COUPLING	Electro-structural Electro-thermal Thermo-mechanical Thermal-electric-structural Piezoelectric Piezoresistive Prestressed modal analysis Stress stiffening	Moving boundary using ALE technique Squeeze-film damping
DEVICES	Cantilever beams Comb drives Resonators Micromirrors Thermomechanical actuators Inertial sensors Pressure sensors	Mechanical pumps and valves

THE MEMS MODULE STUDY CAPABILITIES BY PHYSICS INTERFACE

This section lists the physics interfaces, the physical quantities they solve for, and the standard abbreviation each one uses. The physical quantities in these physics interfaces are:

- The *structural displacements* or the *velocity components* u, v, and w
- The *film pressure variation*, p_f
- The *electric scalar potential*, V
- The *concentration*, c

Table 1-2 lists the physics interfaces specific to MEMS modeling and this module.

	When using the axisymmetric physics interfaces, the horizontal axis represents the r direction and the vertical axis the z direction. The geometry in the right half plane must be created, that is, only for positive r.

|	
	Studies and Solvers in the *COMSOL Multiphysics Reference Manual*

TABLE 1-2: MEMS MODULE INTERFACE DEPENDENT VARIABLES AND PRESET STUDY OPTIONS

PHYSICS INTERFACE	NAME	DEPENDENT VARIABLES	STATIONARY	TIME DEPENDENT	TIME-DEPENDENT MODAL	EIGENFREQUENCY	SMALL-SIGNAL ANALYSIS, FREQUENCY DOMAIN	FREQUENCY DOMAIN	FREQUENCY-DOMAIN MODAL	PRESTRESSED ANALYSIS, EIGENFREQUENCY	PRESTRESSED ANALYSIS, FREQUENCY. DOMAIN
AC/DC											
Electrostatics	es	V	√	√							
Electric Currents	ec	V	√								
Electrical Circuit	cir	none	√	√					√		
FLUID FLOW											

TABLE 1-2: MEMS MODULE INTERFACE DEPENDENT VARIABLES AND PRESET STUDY OPTIONS

PHYSICS INTERFACE	NAME	DEPENDENT VARIABLES	STATIONARY	TIME DEPENDENT	TIME-DEPENDENT MODAL	EIGENFREQUENCY	SMALL-SIGNAL ANALYSIS, FREQUENCY DOMAIN	FREQUENCY DOMAIN	FREQUENCY-DOMAIN MODAL	PRESTRESSED ANALYSIS, EIGENFREQUENCY	PRESTRESSED ANALYSIS, FREQUENCY. DOMAIN
Thin-Film Flow	tff	p_f	√	√		√		√			
Thin-Film Flow, Domain	tff	p_f	√	√		√		√			
Fluid-Structure Interaction	—	u, v, w p	√	√							
Fluid-Structure Interaction, Fixed Geometry	—	u, v, w p	√	√							
STRUCTURAL MECHANICS											
Solid Mechanics	solid	u, v, w	√	√	√	√	√	√	√	√	√
Piezoelectricity	—	u, v, w V	√	√	√	√	√	√	√	√	√
Thermal Stress	—	u, v, w T	√	√	√					√	√
Thermoelasticity	te	u, v, w T		√		√		√			
Joule Heating and Thermal Expansion	—	$u, v, w, T,$ V	√	√	√		√			√	√
Electromechanics	emi	u, v, w V	√	√						√	√
Piezoresistivity, Domain Currents	pzrd	u, v, w V	√	√	√		√	√	√	√	√

TABLE 1-2: MEMS MODULE INTERFACE DEPENDENT VARIABLES AND PRESET STUDY OPTIONS

PHYSICS INTERFACE	NAME	DEPENDENT VARIABLES	PRESET STUDY OPTIONS								
			STATIONARY	TIME DEPENDENT	TIME-DEPENDENT MODAL	EIGENFREQUENCY	SMALL-SIGNAL ANALYSIS, FREQUENCY DOMAIN	FREQUENCY DOMAIN	FREQUENCY-DOMAIN MODAL	PRESTRESSED ANALYSIS, EIGENFREQUENCY	PRESTRESSED ANALYSIS, FREQUENCY. DOMAIN
Piezoresistivity, Boundary Currents	pzrb	u, v, w V	√	√	√		√	√	√	√	√
Piezoresistivity, Shell	pzrs	u, v, w V, ar	√	√	√		√	√	√	√	√

Where Do I Access the Documentation and Application Libraries?

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

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

THE DOCUMENTATION AND ONLINE HELP

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

Opening Topic-Based Help

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

to highlight a node or window, then press F1 to open the Help window, which then displays information about that feature (or click a node in the Model Builder followed by the **Help** button (?). This is called *topic-based* (or *context) help*.

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

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

Opening the Documentation Window

Win	To open the **Documentation** window: • Press Ctrl+F1. • From the **File** menu, select **Help>Documentation** (▮).

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

THE APPLICATION LIBRARIES WINDOW

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

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

🔍	The Application Libraries Window in the *COMSOL Multiphysics Reference Manual*.

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

Win	From the **File** menu, select **Application Libraries**. To include the latest versions of model examples, from the **File>Help** menu, select (▥) **Update COMSOL Application Library**.

Mac Linux	Select **Application Libraries** from the main **File** or **Windows** menus. To include the latest versions of model examples, from the **Help** menu, select (▥) **Update COMSOL Application Library**.

CONTACTING COMSOL BY EMAIL

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

COMSOL ACCESS AND TECHNICAL SUPPORT

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

by email. You can also access technical support, software updates, license information, and other resources by registering for a COMSOL Access account.

COMSOL ONLINE RESOURCES

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

Overview of the User's Guide

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

> As detailed in the section Where Do I Access the Documentation and Application Libraries? this information can also be searched from the COMSOL Multiphysics software **Help** menu.

TABLE OF CONTENTS, GLOSSARY, AND INDEX

Use the Contents, Glossary of Terms, and Index to help you navigate through this guide.

MEMS MODELING

Creating and Analyzing MEMS Models familiarizes you with modeling procedures useful when working with this module and MEMS applications such as microactuators and microsensors. Because the module is fully integrated with COMSOL Multiphysics, the design and analysis process is similar to the one you use in the base environment. The MEMS-specific modeling aspects should prove valuable during the modeling phase. There is a also short section (with model examples) about the materials databases available with this module.

THE AC/DC BRANCH

AC/DC Interfaces chapter describes the physics interfaces for modeling conducting and nonconducting materials, including the enhanced versions of the Electrostatics and Electric Currents interfaces. It also has all the information about the Electrical Circuit interface. The Magnetic Fields interface is also included as part of the COMSOL Multiphysics base package, but this is detailed in the *COMSOL Multiphysics Reference Manual*.

THE FLUID FLOW BRANCH

The types of momentum transport that you can simulate include laminar and thin-film flow. The Laminar Flow interface is available with the base license and is described in the *COMSOL Multiphysics Reference Manual*. Fluid Flow Interfaces chapter describes the Thin-Film Flow and Fluid-Structure Interaction interfaces.

The Thin-Film Flow Interfaces (in the *CFD Module User's Guide*) can be added either singularly or in combination with other physics interfaces such as mass and energy transfer. The Thin-Film Flow interface is for 2D, 2D axisymmetric and 3D models, and the Thin-Film Flow, Domain interface is for 2D models.

The Fluid-Solid Interaction Interface (in the *Structural Mechanics Module User's Guide*) is an interface found under the Fluid Flow branch. It has the equations and physics for fluid-structure interaction, solving for the displacements, fluid velocity, and fluid pressure. You have access to all functionality from the Solid Mechanics interface and Single-Phase Flow interfaces for modeling the solid and fluid domains. From the Laminar Flow (the default) and Solid Mechanics submenus on the solid and boundary level you can access materials, sources, loads, and boundary conditions for the individual physics interfaces.

Fluid-Structure Interaction, Fixed Geometry multiphysics coupling (in the *Structural Mechanics Module User's Guide*) can be used to model phenomena where a fluid and a deformable solid structure affect each other. Both the fluid loading on the structure and the structural velocity transmission to the fluid can be taken into account. The interface models situations where the displacements of the solid are assumed to be small enough for the geometry of the fluid domain to be considered as fixed during the interaction.

THE STRUCTURAL MECHANICS BRANCH

Structural Mechanics Interfaces chapter describes the Solid Mechanics, Piezoelectricity, Thermal Stress, Joule Heating and Thermal Expansion, and Electromechanics interfaces.

The Solid Mechanics Interface
The Electromechanics Interface has the equations and physics features for stress analysis and general linear solid mechanics, solving for the displacements.

The Piezoelectricity Interface
The Piezoelectricity Interface, Solid combines piezoelectricity with solid mechanics and electrostatics for modeling of piezoelectric devices where all or some of the domains contain a piezoelectric material. The piezoelectric coupling can be on stress-charge or strain-charge form.

The Thermal Stress Interface
The Electromechanics Interface combines Solid Mechanics and Heat Transfer for modeling stress analysis and general linear and nonlinear solid mechanics by solving for the displacements.

The Joule Heating and Thermal Expansion Interface

The Joule Heating and Thermal Expansion Interface combines solid mechanics using a thermal linear elastic material with an electromagnetic Joule heating model. This is a multiphysics combination of solid mechanics, electric currents, and heat transfer for modeling of, for example, thermoelectromechanical (TEM) applications.

The Electromechanics Interface

The Electromechanics Interface combines solid mechanics and electrostatics with a moving mesh to model the deformation of electrostatically actuated structures. The physics interface is also compatible with piezoelectric materials.

The Thermoelasticity Interface

The Thermoelasticity Interface combines the Solid Mechanics and Thermal Stress interfaces, together with the thermoelastic coupling terms. These coupling terms result in local cooling of material under tension and heating of material that is compressed. The resulting irreversible heat transfer between warm and cool regions of the solid produces mechanical losses, which can be important, particularly for small structures.

THE PIEZORESISTIVITY BRANCH

These physics interfaces are appropriate for modeling piezoresistivity under various assumptions.

The Piezoresistivity, Domain Currents Interface is appropriate for situations when the thickness of the conducting and piezoresistive layers are both resolved by the mesh.

The Piezoresistivity, Boundary Currents Interface should be used when the thicknesses of the conducting and piezoresistive layers are much smaller than those of the structural layers (this is often the case in practice).

The Piezoresistivity, Shell Interface is used when the structural layer is thin enough to be treated by the shell interface, but the conducting and piezoresistive layers are still much thinner than the structural layers.

2

Creating and Analyzing MEMS Models

This chapter gives an overview of the physics interfaces and features available for modeling MEMS devices in COMSOL Multiphysics®.

In this chapter:

- Physics and Scaling for MEMS Devices
- General Tools for MEMS Modeling
- Structural Mechanics
- Electrostatics
- Piezoresistivity
- Thermal Effects in MEMS Devices
- Microfluidics
- Connecting to Electrical Circuits
- SPICE Import and Export

Physics and Scaling for MEMS Devices

Microelectromechanical systems (MEMS) is a term that covers a broad range of manufactured mechanical components characterized by a small size. Many of these devices are fabricated with micron-scale dimension and represent mixed mechanical/electrical systems, although the term now encompasses systems that do not necessarily contain electrical parts, such as microfluidic devices. Thousands of MEMS devices can be batch fabricated on a single wafer, using technologies derived from the semiconductor industry. As a result of the smaller size and reduced manufacturing costs, MEMS-based equivalents of macroscopic devices such as accelerometers have been able to penetrate new markets. Additionally, MEMS technology has enabled entirely new devices to be created, such as the micromirror arrays that enable Texas Instruments' DLP projectors.

> The Physics Interfaces and Building a COMSOL Multiphysics Model in the *COMSOL Multiphysics Reference Manual*

The smaller size of MEMS devices has important effects on the physical processes that govern their operation (Ref. 1). In general, as the length scale (L) of the device is reduced, the scaling of a physical effect with respect to L determines its relative importance (see Table 2-1). The inertial force required to produce a fixed acceleration of a solid body scales volumetrically as L^3. The scaling of other forces in comparison to this inertial force has important consequences for MEMS devices. For example, the effective spring constant for a body scales as L^1. The spring stiffness therefore decreases much more slowly than the system mass as the size of the system is reduced, resulting in higher resonant frequencies for smaller devices (resonant frequency scales as L^{-1}). This means that micromechanical systems typically have higher operating frequencies and faster response times than macroscopic systems.

> See Structural Mechanics for more information about modeling the mechanics of MEMS devices.

Electrostatic forces scale favorably as the device dimensions are reduced (for example, the force between parallel plates with a fixed applied voltage scales as L^0) (see Table 2-1). Additionally, electrostatic actuators consume no DC power and can be

manufactured using processes that are compatible with standard semiconductor foundries. In comparison, magnetic actuators scale less favorably (for example, the force between wires carrying a fixed current density scales as L^4) and can involve materials that are incompatible with standard semiconductor processing. In some cases magnetic actuators or sensors require a DC bias current, which increases power consumption. These factors explain the prevalence of electrostatic actuation and sensing and the absence of magnetically actuated commercial MEMS devices.

TABLE 2-1: SCALING OF VARIOUS PHYSICAL PHENOMENA WITH DECREASING LENGTH (L) AND THE CONSEQUENCES FOR MEMS DEVICES

PROPERTY	LENGTH SCALING	CONSEQUENCES
Volume	L^3	Surface related properties increase in importance relative to volumetric properties, Surface area to volume ratios are high.
Area	L^2	
Inertial Forces	L^3	How other forces scale with respect to inertial forces determines how effective device actuation is.
Spring Constant	L^1	Spring constant to mass ratio is reduced. Higher resonant frequencies, faster response times.
Resonant Frequency	L^{-1}	
Electrostatic force (parallel plates, constant voltage)	L^0	Electrostatic actuation becomes more efficient and is widely used in MEMS devices.
Magnetic force (parallel wires, constant current density)	L^4	Magnetic actuation scales poorly; there are no commercial magnetically actuated devices.
Piezoelectric force (constant voltage)	L^1	Piezoelectric actuation is efficient (manufacturing difficulties have hindered commercial adoption).
Thermal force (constant temperature difference)	L^2	Thermal actuation is efficient and fast. However, power consumption is typically greater than other methods.
Thermal time scale	L^2	
Squeeze film damping force	L^1	Squeeze film damping becomes relatively more important.

Electrostatically actuated and sensed MEMS devices dominate the market for inertial sensors (usually employed in the automobile and consumer electronics industries). Magnetic MEMS devices can also be modeled in COMSOL Multiphysics, but since

these are less common, the AC/DC Module is required in addition to the MEMS Module.

> 🔍 See Electrostatics for information about how to model electrostatic phenomena in MEMS devices.

Piezoelectric forces also scale well as the device dimension is reduced (the force produced by a constant applied voltage scales as L^1). Furthermore, piezoelectric sensors and actuators are predominantly linear and do not consume DC power in operation. Piezoelectrics are more difficult to integrate with standard semiconductor processes, but significant progress has been made with commercial successes in the market (for example, Avago Technologies (formerly Agilent/HP) FBAR filters). High frequency FBAR resonators fabricated from aluminum nitride thin films on silicon wafers are now widely used as filters used in consumer devices such as mobile phones.

> 🔍 Modeling Piezoelectric Problems

Thermal forces scale as L^2, assuming that the forces are generated by a fixed temperature change. This scaling is still favorable in comparison to inertial forces, and the thermal time scale also scales well (as L^2), making thermal actuators faster on the microscale (although thermal actuators are typically slower than capacitive or piezoelectric actuators). Thermal actuators are also easy to integrate with semiconductor processes although they usually consume large amounts of power and thus have had a limited commercial applicability. Thermal effects play an important role in the manufacture of many commercial MEMS technologies with thermal stresses in deposited thin films being critical for many applications.

> 🔍 See Thermal Effects in MEMS Devices, which discusses thermal actuation and stresses in more detail.

The interaction of MEMS devices with a fluid leads to a range of microfluidic devices. COMSOL also has the Microfluidics Module, which is geared to model microfluidic devices, but some functionality to model fluids is included with the MEMS Module. In particular, the interaction of moving components with a fluid can be modeled using the Fluid-Structure Interaction Interface. In addition to modeling microfluidic actuators, this multiphysics interface can be used to model the damping of moving

structures in an air ambient. Damping in MEMS devices is often more conveniently modeled by treating the air as a thin layer and solving the Reynolds equations (or a modified form). These equations are also included in this module. The damping forces scale as L^1 and are often important for MEMS devices, frequently leading to a requirement for vacuum packaging.

Q | Microfluidics

This discussion has focused on actuators; however, many of the actuators discussed can also be used as sensors. Many MEMS devices (for example, gyroscopes and resonant sensors) require both actuators and sensors to be integrated into the same device. Electrostatic, magnetic, piezoelectric, and even thermal devices have all been employed as MEMS sensors. Another commonly used sensing mechanism found in commercial MEMS devices is that of piezoresistivity. The piezoresistive effect refers to the change in a material's conductivity that occurs in response to an applied stress. Piezoresistors are relatively easy to manufacture using common semiconductor processes and are inherently shielded from electromagnetic interference, and it is usually straightforward to implement electronics to interface with them. Pressure sensors based on the piezoresistive effect were some of the first MEMS devices to be mass produced. Piezoresistive devices do, however, usually consume more power and generate more electrical noise than capacitive sensors, which are displacing piezoresistive devices in some applications.

Q | Piezoresistivity

General Tools for MEMS Modeling

Several tools are required to model MEMS devices in addition to the application-specific physics interfaces. These are discussed in the following sections:

- Infinite Elements and Perfectly Matched Layers
- Electrical Circuits
- Reduced-Order Modeling and Equivalent Circuits
- Importing Mask Data
- About the Material Libraries for the MEMS Module

Infinite Elements and Perfectly Matched Layers

Infinite elements and *perfectly matched layers* (PMLs) are frequently required when modeling MEMS devices.

- Infinite elements are used in stationary or transient computations enabling a (finite sized) region of the model to represent a region stretching out to infinity. This applies to problems where the solution varies without oscillation in the infinite domain, a typical MEMS application is to compute the capacitance of a structure in a surrounding air ambient.
- Perfectly matched layers enable outgoing waves in a frequency domain model to be perfectly adsorbed. A typical MEMS application is to use PMLs to compute the anchor losses for a vibrating structure.

🔍	Infinite Elements, Perfectly Matched Layers, and Absorbing Layers in the *COMSOL Multiphysics Reference Manual*

Electrical Circuits

Many MEMS devices include accompanying electrical circuits, either integrated on the same chip or, increasingly, through connections to an adjacent chip that is packaged with the device. COMSOL Multiphysics includes tools to connect physical finite

element models to electrical circuits as well as an Electric Circuits interface to model the circuits themselves.

> The Connecting to Electrical Circuits section discusses how to use COMSOL Multiphysics in conjunction with external circuits.

Reduced-Order Modeling and Equivalent Circuits

Reduced-order modeling seeks to reduce the number of degrees of freedom in a physical model, whilst still retaining the essential physics. It is frequently used in the design of resonant MEMS devices, which are usually designed to operate with the system vibrating in a particular mode. For a lightly damped resonant system driven at one of its resonant frequencies, it is reasonable to consider only the contributions to the system of a small number (m) of modes within the signal bandwidth. In some cases a single mode is sufficient. A system with n degrees of freedom has mass, stiffness, and damping matrices of size n-by-n. A reduced-order representation of the system considering m modes has size m-by-m. The reduction in complexity of the system, and the computational speed up is therefore significant when $m \ll n$. Determining a suitable reduced-order model of a complex system is a common simulation goal for MEMS designers. Frequently this is implemented as an equivalent circuit that can easily be integrated with SPICE models of the system as a whole. This section describes the theory of the reduced-order system and gives guidelines on how to obtain reduced-order models from a COMSOL Multiphysics model.

THE MODAL COORDINATE SYSTEM

Consider a mechanical system, with n degrees of freedom, described by an equation of the form

$$M\ddot{\mathbf{u}} + D\dot{\mathbf{u}} + K\mathbf{u} = \mathbf{F} \tag{2-1}$$

where $\mathbf{u}$ is the displacement vector (size: n-by-1), K is the stiffness matrix (size: n-by-n), D is the damping matrix (size: n-by-n), and M is the mass matrix (size: n-by-n). In the frequency domain the problem takes the form

$$-\omega^2 M\mathbf{u}_0 + i\omega D\mathbf{u}_0 + K\mathbf{u}_0 = \mathbf{F}$$

where $\mathbf{u} = \mathbf{u}_0 e^{i\omega t}$.

Initially consider the system in the absence of damping and forces. The undamped system has n eigenvalues ω_i, which satisfy the equation

$$K\hat{\mathbf{u}}_i = \omega_i^2 M\hat{\mathbf{u}}_i \qquad\qquad (2\text{-}2)$$

These eigenvectors are orthogonal with respect to M, as can be seen from the following construction, based on Equation 2-2:

$$\hat{\mathbf{u}}_j^T K \hat{\mathbf{u}}_i - \omega_i^2 \hat{\mathbf{u}}_j^T M \hat{\mathbf{u}}_i = 0$$

$$\hat{\mathbf{u}}_i^T K \hat{\mathbf{u}}_j - \omega_j^2 \hat{\mathbf{u}}_i^T M \hat{\mathbf{u}}_j = 0$$

$$(\hat{\mathbf{u}}_i^T K \hat{\mathbf{u}}_j - \omega_j^2 \hat{\mathbf{u}}_i^T M \hat{\mathbf{u}}_j)^T = \hat{\mathbf{u}}_j^T K \hat{\mathbf{u}}_i - \omega_j^2 \hat{\mathbf{u}}_j^T M \hat{\mathbf{u}}_i = 0$$

using the results $M^T = M$, and $K^T = K$, which apply for physical reasons. Subtracting the first equation from the third gives:

$$-(\omega_j^2 - \omega_i^2)\hat{\mathbf{u}}_j^T M \hat{\mathbf{u}}_i = 0$$

so provided that the eigenfrequency are different:

$$\hat{\mathbf{u}}_j^T M \hat{\mathbf{u}}_i = 0 \qquad i \neq j, \ \omega_i \neq \omega_j \qquad\qquad (2\text{-}3)$$

Note that the above equations also imply

$$\hat{\mathbf{u}}_j^T K \hat{\mathbf{u}}_i = 0 \qquad i \neq j, \ \omega_i \neq \omega_j \qquad\qquad (2\text{-}4)$$

Next the following n-by-n matrix is constructed, with columns taken from the n eigenvectors:

$$U = \begin{bmatrix} \hat{\mathbf{u}}_1, \ \hat{\mathbf{u}}_2 \ \dots \ \hat{\mathbf{u}}_n \end{bmatrix}$$

Then consider the following matrix:

$$U^T M U = \begin{bmatrix} \hat{\mathbf{u}}_1^T M \hat{\mathbf{u}}_1 & \hat{\mathbf{u}}_1^T M \hat{\mathbf{u}}_2 & & & \\ \hat{\mathbf{u}}_2^T M \hat{\mathbf{u}}_1 & \hat{\mathbf{u}}_2^T M \hat{\mathbf{u}}_2 & & & \\ & & \ddots & & \\ & & & \hat{\mathbf{u}}_{n-1}^T M \hat{\mathbf{u}}_{n-1} & \hat{\mathbf{u}}_{n-1}^T M \hat{\mathbf{u}}_n \\ & & & \hat{\mathbf{u}}_n^T M \hat{\mathbf{u}}_{n-1} & \hat{\mathbf{u}}_n^T M \hat{\mathbf{u}}_n \end{bmatrix}$$

From Equation 2-3 it is clear that this is a diagonal matrix. Similarly from Equation 2-4 it is clear that $U^T K U$ is also diagonal.

From the properties of the eigenvectors it is possible to expand any function in terms of the eigenvectors. Thus the displacement $\mathbf{u}$ can be written as:

$$\mathbf{u} = \sum_{i=1}^{n} a_i \hat{\mathbf{u}}_i$$

This equation can also be expressed in the form:

$$\mathbf{u} = U\mathbf{a} \tag{2-5}$$

where $\mathbf{a}$ is a column vector containing the coefficients a_i as rows. In general $\mathbf{a}$ is time dependent.

Now consider the original equation: Equation 2-1. First substitute for $\mathbf{u}$ using Equation 2-5. Then transform the equation to the *modal coordinate system* by premultiplying by U^T. This gives:

$$U^T M U \ddot{\mathbf{a}} + U^T D U \dot{\mathbf{a}} + U^T K U \mathbf{a} = U^T \mathbf{F} \tag{2-6}$$

It has already been established that the matrices $U^T M U$ and $U^T K U$ are diagonal, and frequently a damping model is chosen that results in a diagonal damping matrix. For example, in Rayleigh damping $D = \alpha M + \beta K$, where α and β are constants. Alternatively, a damping ratio, ξ_i, is assigned to each mode.

EIGENVALUE SCALING AND EQUIVALENT ELECTRICAL AND MECHANICAL SYSTEMS

The precise form of Equation 2-6 is determined by the normalization adopted for the eigenfunctions. In structural applications the eigenfunctions are often normalized such that $U^T M U = I$. This is referred to as *mass matrix* scaling in the eigenvalue solver. In this case Equation 2-2 gives

$$\hat{\mathbf{u}}_i^T K \hat{\mathbf{u}}_i = \omega_i^2 \hat{\mathbf{u}}_i^T M \hat{\mathbf{u}}_i = \omega_i^2$$

so that

$$U^T K U = \text{diag}(\omega_i^2)$$

where $\text{diag}(\omega_i^2)$ is the diagonal matrix with diagonal elements ω_i^2. Similarly, if damping ratios for each mode are defined, the damping matrix can be expressed in the form

$$U^T D U = \text{diag}(2\xi_i \omega_i)$$

Thus if *mass matrix* scaling is used Equation 2-6 takes the form

$$\ddot{\mathbf{a}} + \text{diag}(2\xi_i \omega_i)\dot{\mathbf{a}} + \text{diag}(\omega_i^2)\mathbf{a} = U^T \mathbf{F} \qquad (2\text{-}7)$$

In MEMS applications it is common to scale the eigenvectors so that the point of maximum displacement has unit displacement magnitude. This is referred to as *max* scaling in the eigenvalue solver. For an individual mode this scaling has a simple physical interpretation — the corresponding component of $\mathbf{a}$, a_i, is the amplitude of the ith mode, measured at the point of maximum displacement, when the mode is driven by the force $\mathbf{F}$. In this case Equation 2-6 takes the form

$$\text{diag}(m_{\text{eff}, i})\ddot{\mathbf{a}} + \text{diag}(c_{\text{eff}, i})\dot{\mathbf{a}} + \text{diag}(k_{\text{eff}, i})\mathbf{a} = U^T \mathbf{F} \qquad (2\text{-}8)$$

Here $m_{\text{eff},i}$ is the effective mass of the ith mode, $c_{\text{eff},i} = 2m_{\text{eff},i}\xi_i\omega_i$ is the effective damping parameter for the mode, and $k_{\text{eff},i}$ is the effective spring constant. Each element of the vector $U^T \mathbf{F}$ gives the force component that acts on each of the respective modes.

For electromechanical systems it is common to express the force term, $U^T \mathbf{F}$, in terms of a driving voltage, V, applied to a terminal in the system. Given this:

$$U^T \mathbf{F} = \eta V$$

here the vector η has components η_i which correspond to the *electromechanical coupling factor* for each of the modes in the system. Equation 2-8 now takes the form

$$\text{diag}(m_{\text{eff}, i})\ddot{\mathbf{a}} + \text{diag}(c_{\text{eff}, i})\dot{\mathbf{a}} + \text{diag}(k_{\text{eff}, i})\mathbf{a} = \eta V \qquad (2\text{-}9)$$

To obtain the electromechanical coupling coefficients for a set of modes, it is possible to apply a 1 V signal to the system and then compute the induced forces: $U^T \mathbf{F} = \eta$. Reciprocally, energy conserving transducers also have the property that the current that flows into the transducer when it is operating in a particular mode ($i_{mot,i}$) is related to the displacement through the equation

$$i_{mot,\, i} = \eta_i \dot{a}_i$$

So Equation 2-9 can be rewritten in the form

$$\mathrm{diag}\left(\frac{m_{\mathrm{eff},\, i}}{\eta_i^2}\right)\frac{d\mathbf{i}}{dt} + \mathrm{diag}\left(\frac{c_{\mathrm{eff},\, i}}{\eta_i^2}\right)\mathbf{i} + \mathrm{diag}\left(\frac{k_{\mathrm{eff},\, i}}{\eta_i^2}\right)\int \mathbf{i}\, dt = \mathbf{V} \tag{2-10}$$

For each mode, the terms on the left of Equation 2-10 can be identified with a resistor capacitor and inductor, such that the mechanical system behaves like a set of parallel series *LCR* circuits. The equivalent modal resistances, capacitances, and inductances are given by:

$$L_i = \frac{m_{\mathrm{eff},\, i}}{\eta_i^2}$$

$$R_i = \frac{c_{\mathrm{eff},\, i}}{\eta_i^2}$$

$$C_i = \frac{\eta_i^2}{k_{\mathrm{eff},\, i}}$$

For a given mode, these quantities are frequently referred to as the *motional inductance*, *resistance*, and *capacitance*.

REDUCED-ORDER MODELS

The preceding discussion did not consider how to reduce the number of degrees of freedom in the system. For systems in which the vector $U^T \mathbf{F}$ has only a few significant components (for example, components $i = 1, \ldots, m$ where $m \ll n$) the following approximation can be made:

$$\mathbf{u} \approx \sum_{i=1}^{m} a_i \hat{\mathbf{u}}_i$$

The expression for **u** in matrix becomes:

$$\mathbf{u} = U'\mathbf{a}'$$

where U' is now an m-by-n and $\mathbf{a}'$ is a vector with m components. The equation system in modal coordinates now takes the form

$$U'^{T}MU'\ddot{\mathbf{a}}' + U'^{T}DU'\dot{\mathbf{a}}' + U'^{T}KU'\mathbf{a}' = U'^{T}\mathbf{F} \qquad (2\text{-}11)$$

The matrices $U'^{T}MU'$, $U'^{T}DU'$, and $U'^{T}KU'$ now have dimensions m-by-m. Similarly the vector $U'^{T}\mathbf{F}$ has m components. This results in a significant reduction in the system complexity.

REDUCED-ORDER MODELS WITH PHYSICAL DAMPING

If physically relevant damping is present in the system, the above theory must be modified as the damping matrix is no longer diagonal in the modal coordinate system. COMSOL can still handle this case as the modal solver does not make the assumption that any of the matrices are diagonal. In this case the eigenvalues become complex and the eigenvectors split into right and left eigenvectors. The right eigenvectors U^{r} are solutions of the equation:

$$-\omega_{i}^{2}M\hat{\mathbf{u}}_{r,i} + i\omega_{i}D\hat{\mathbf{u}}_{r,i} + K\hat{\mathbf{u}}_{r,i} = \mathbf{F}$$

As in the previous section, for a reduced set of modes, it is assumed that:

$$\mathbf{u} = U'_{r}\mathbf{a}'$$

where U'_{r} is the n-by-m matrix containing the right eigenvectors chosen for the modal analysis. Once again $\mathbf{a}'$ is a vector with m components. The system in modal coordinates takes the form

$$U'_{l}{}^{T}MU'_{r}\ddot{\mathbf{a}}' + U'_{l}{}^{T}DU'_{r}\dot{\mathbf{a}}' + U'_{l}{}^{T}KU'_{r}\mathbf{a}' = U'_{l}{}^{T}\mathbf{F}$$

where U'_{l} is the n-by-m matrix containing the left eigenvectors chosen for the modal analysis.

The matrices $U'_{l}{}^{T}MU'_{r}$, $U'_{l}{}^{T}DU'_{r}$, and $U'_{l}{}^{T}KU'_{r}$ are no longer necessarily diagonal. The modal solver accepts any linearly independent set of vectors to project the solution vector and equations onto and constructs the reduced-order system accordingly.

The **Model Reduction** and **Modal Reduced-Order Model** study steps have the property that they can assemble the modal matrices and make them available for output. In the **Model Reduction** node, the **Store reduced matrices** check box must be selected.

After the model has solved, right-click the **Results>Derived Values** node and select **System Matrices**. In the output section choose the **Matrix** to display in the list. The mass matrix corresponds to the matrix $U'_l{}^T M U'_r$ the stiffness matrix corresponds to $U'_l{}^T K U'_r$, and the damping matrix corresponds to $U'_l{}^T D U'_r$. The vector $U'_l{}^T \mathbf{F}$ is available as the load vector. These matrices are given in a format that respects the normalization of the preceding **Eigenvalue Solver**. To change this select the **Eigenvalue Solver** node, and change the **Scaling of Eigenvectors** setting under the **Output** section. Use the **Max** setting if an equivalent Mass-Spring-Damper system is required, in which case the modal amplitude corresponds to the maximum displacement of the mode.

To convert this into an equivalent circuit representation, see the discussion in the section Eigenvalue scaling and Equivalent Electrical and Mechanical Systems.

Importing Mask Data

The semiconductor processes used to manufacture MEMS devices are usually specified layer by layer through the definition of a mask set. It is often convenient to use these mask sets when building models of the 3D devices in COMSOL Multiphysics.

💡	Use the ECAD Import Module to import mask data into COMSOL Multiphysics. See the *ECAD Import Module User's Guide* or go to https://www.comsol.com/ecad-import-module/ for more information about this and other COMSOL products.

About the Material Libraries for the MEMS Module

The MEMS Module includes these material libraries:

- *MEMS*, an extended solid materials library with metals, semiconductors, insulators, and polymers common in MEMS devices.

- *Piezoelectric*, with over 20 common piezoelectric materials. The materials include temperature-dependent fluid dynamic and thermal properties.

- *Piezoresistivity*, with four materials each with seven material properties.

In the *COMSOL Multiphysics Reference Manual:*

- MEMS Material Library
- Piezoelectric Materials Library
- Piezoresistivity Materials Library
- Materials

For an example of the MEMS materials database, see *Microresistor Beam*: Application Library path **MEMS_Module/Actuators/microresistor_beam**.

For examples of the Piezoelectric materials database, see:

- *Piezoelectric Shear-Actuated Beam*: Application Library path
  MEMS_Module/Piezoelectric_Devices/shear_bender
- *Piezoceramic Tube*: Application Library path
  MEMS_Module/Piezoelectric_Devices/piezoceramic_tube

For examples of the Piezoresistivity materials database, see:

- *Piezoresistive Pressure Sensor*: Application Library path
  MEMS_Module/Sensors/piezoresistive_pressure_sensor
- *Multiphysics Interfaces and Couplings*: Application Library path
  MEMS_Module/Sensors/piezoresistive_pressure_sensor_shell

Structural Mechanics

Many of the COMSOL Multiphysics structural mechanics physics interfaces are available in the MEMS Module and are found under the **Structural Mechanics** branch (⬚) when adding a physics interface.

Many important aspects of the creation of structural mechanics models are covered in the chapter Structural Mechanics Modeling in the *Structural Mechanics Module User's Guide*. The following sections are of special importance for MEMS modeling;

- Selecting the Physics Interface
- Applying Loads
- Defining Constraints
- Calculating Reaction Forces
- Modeling Piezoelectric Problems
- Mechanical Damping and Losses
- Modeling Geometric Nonlinearity
- Contact Modeling
- Springs and Dampers
- Defining Multiphysics Models
- Thermally Coupled Problems
- Solver Settings for Structural Mechanics

🔍	In the *Structural Mechanics Module User's Guide* - The Solid Mechanics Interface - Structural Mechanics Theory

---

❗	The links to external guides, such as the *Structural Mechanics Module User's Guide*, do not work in the PDF, only from the online help in COMSOL Multiphysics.

Damping and Losses

For frequency domain (eigenfrequency or frequency domain studies) and transient simulations it is often necessary to include damping in the model. This is particularly important for resonant MEMS devices.

Within COMSOL Multiphysics, damping can be modeled explicitly by coupling the relevant physics interfaces into the model. Examples of such couplings include:

- Modeling the movement of the structure through a fluid directly by means of a complete fluid-structure interaction model. This can be done using the Fluid-Structure Interaction interface.

- Modeling thin films of a fluid using The Thin-Film Flow Interfaces (in the *CFD Module User's Guide*), or by means of the thin film damping boundary condition, integrated into the structural mechanics physics interfaces.

- Modeling thermoelastic damping explicitly by including heat transfer and the relevant thermodynamics in the model. See The Thermoelasticity Interface.

- Modeling acoustic losses through anchor structures in the frequency domain using a perfectly matched layer.

Infinite Elements, Perfectly Matched Layers, and Absorbing Layers in the *COMSOL Multiphysics Reference Manual*

- Modeling losses through anchor structures in the time domain using a spring damper system.

In the *Structural Mechanics Module User's Guide*

- The Fluid-Solid Interaction Interface
- Mechanical Damping and Losses
- Springs and Dampers

Electrostatics

Most MEMS devices meet the electroquasistatic criterion (Ref. 2) in which the wavelength of the applied fields is substantially larger than the modeled device. The electric field can therefore be treated as quasi static and The Electrostatics Interface or The Electric Currents Interface can be used.

In this section:

- Electrostatics
- Electric Currents in Conductive Media
- Lumped Parameters

Electrostatics

The Electrostatics Interface solves for the electric potential V using the equation

$$-\nabla \cdot (\varepsilon \nabla V) = \rho$$

For most MEMS devices the details of the charge distribution, ρ, inside material are unimportant and the charge usually resides only on surfaces held at fixed potentials. The Electric Potential, Ground, and Terminal boundary conditions are usually used to fix the potential on the surfaces of a MEMS device. At symmetry boundaries the Zero Charge boundary condition applies. To represent infinite regions adjacent to the model a common approach is to use infinite elements. Note that it is possible to model electric circuits associated with a terminal, as described in the Connecting to Electrical Circuits section.

> ⊙ Infinite Elements, Perfectly Matched Layers, and Absorbing Layers in the *COMSOL Multiphysics Reference Manual*

ELECTROSTATIC FORCES

The Maxwell stress tensor is a general method for calculating electromagnetic forces and torques. Force and torque calculations using Maxwell's stress tensor are available in the Electrostatics interface. In electrostatics the force is calculated by integrating

$$\mathbf{n}_1 T_2 = -\frac{1}{2}\mathbf{n}_1(\mathbf{E} \cdot \mathbf{D}) + (\mathbf{n}_1 \cdot \mathbf{E})\mathbf{D}^T \tag{2-12}$$

on the surface of the object that the force acts on where $\mathbf{E}$ is the electric field, $\mathbf{D}$ the electric displacement, and $\mathbf{n}_1$ the outward normal from the object.

In practice, electric fields are modeled not only in the air but also inside the solid materials. By entering the electromagnetic force variable name into the domain settings, the force calculations are activated, and the physics interface automatically generates new domain and boundary variables.

	The form of the stress tensor in Equation 2-12 assumes no material electrostriction. Support for isotropic electrostriction is included in The Electromechanics Interface.

CAPACITANCE CALCULATIONS

The capacitance of a system can be computed using the functionality built into the Terminal boundary condition. This is discussed in more detail in the section Lumped Parameters.

Electric Currents in Conductive Media

The Electric Currents Interface uses the equation

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

where σ is the conductivity, $\mathbf{J}^e$ is an externally generated current density, and Q_j is the volume current source, which is usually zero in MEMS applications. Current flow through the device is specified by means of the Normal Current Density and Electric Insulation physics nodes. The Electric Potential, Ground, and Terminal physics nodes are frequently employed to specify constant potential surfaces, which can also act as current sources or sinks.

Conductors in MEMS designs are often thin sheets. For cases in which large aspect ratio differences occur it is often convenient to represent thin layers using a planar approximation.

Use **Terminal** boundary conditions to calculate resistances or conductances of the system. For more details see the Lumped Parameters section.

▓▓▓	*Thin-Film Resistance*: Application Library path **COMSOL_Multiphysics/Electromagnetics/thin_film_resistance**

Lumped Parameters

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

CALCULATING LUMPED PARAMETERS WITH OHM'S LAW

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

There are several available techniques to extract the lumped parameters. Which one to use depends on the physics interface, the parameter of interest, and how the model is solved. The overview of the techniques in this section use a 4-by-4 matrix example for the lumped parameter matrix. This represents a system of at least five electrodes, where four are used as terminals and the rest are grounded, as illustrated in Figure 2-1.
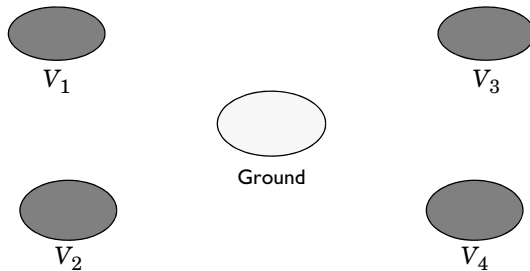
Figure 2-1: A five-electrode system with 4 terminals and one ground electrode.

If a system specifies that all electrodes are terminals, the results are redundant matrix elements. This is better understood by considering a two-electrode system. If both

electrodes are declared as terminals, a 2-by-2 matrix is obtained for the system. This is clearly too many elements because there is only one unique lumped parameter between the terminals. If in addition one or more ground electrodes are declared, the system has three unique electrodes and the lumped parameter matrix becomes a 2-by-2 matrix.

FORCED VOLTAGE

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

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

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

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

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

FIXED CURRENT

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

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

FIXED CHARGE

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

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

CALCULATING LUMPED PARAMETERS USING THE ENERGY METHOD

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

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

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

S-PARAMETERS

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

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

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

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

Now, for high-frequency problems, voltage is not a well-defined entity, and it is necessary to define the scattering parameters in terms of the electric field.

S-Parameter Calculations

The MEMS interfaces have built-in support for S-parameter calculations. In the Electric Currents and Electrostatics interfaces, use the *terminal* boundary feature with the *terminated* setting to approximate a connecting transmission line or a voltage source with a known internal impedance. For a terminal the voltage measurement is always with respect to ground so at least one *ground* feature is also required in the model.

S-Parameter Variables

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

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

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

Port/Terminal Sweeps and Touchstone Export

The, Manual Terminal Sweep Settings section in the Electrostatics interface and the Manual Terminal Sweep Settings section in the Electric Currents interface describe how to cycle through the terminals, compute the entire S-matrix and export it to a Touchstone file.

LUMPED PARAMETER CONVERSION

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

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

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

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

where $\mathbf{L}$ is the inductance, $\mathbf{C}$ is the capacitance, $\mathbf{R}$ is the resistance, and $\mathbf{G}$ is the conductance. $\mathbf{S}$ is the S-parameter. The relations also include the following matrices

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

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

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

where Z_0 is the characteristic impedance.

You can compute conversions between the impedance matrix, $\mathbf{Z}$, the admittance matrix, $\mathbf{Y}$, and the S-parameter matrix S in a results table using the **Settings** window for the **Global Matrix Evaluation** node, which you can add under **Results>Derived Values**.

> 🔍 Global Matrix Evaluation in the *COMSOL Multiphysics Reference Manual*

Piezoresistivity

The *piezoresistive effect* describes the change in a material's conductivity when a stress is applied to the material. Unlike the piezoelectric effect, piezoresistivity is not reversible, so an applied current does not induce a stress (unless other secondary effects are present, such as heating). Piezoresistance is usually associated with semiconductor materials. In semiconductors, piezoresistance results from the strain-induced alteration of the material's band structure and the associated changes in carrier mobility and number density.

Piezoresistivity can be described in terms of a stress-induced change in the resistivity of the material. The relation between the electric field, $\mathbf{E}$, and the current, $\mathbf{J}$, becomes:

$$\mathbf{E} = \rho \cdot \mathbf{J} + \Delta\rho \cdot \mathbf{J}$$

where ρ is the resistivity and $\Delta\rho$ is the induced change in the resistivity. In the general case, ρ and $\Delta\rho$ are both rank-2 tensors (matrices). The change in resistance is related to the stress, σ, (for the piezoresistance form of the equations) or the strain, ε, (for the elastoresistance form of the equations) by the constitutive relationship:

$$\Delta\rho = \Pi \cdot \sigma \qquad \text{(piezoresistance form)}$$
$$\Delta\rho = M \cdot \varepsilon \qquad \text{(elastoresistance form)}$$

(2-13)

where Π is the piezoresistance tensor (SI unit: $\mathrm{Pa}^{-1}\Omega\cdot\mathrm{m}$) and M is the elastoresistance tensor (SI unit: $\Omega\cdot\mathrm{m}$). Note that both of these quantities are material properties. Π and M are in this case rank-4 tensors; however, they can be represented as matrices if the resistivity, stress, and strain are converted to vectors within a reduced subscript notation.

In practice, piezoresistors are fabricated by doping a thin layer of a lightly doped wafer, either using the process of ion implantation or by diffusing dopant into the wafer from its surface at high temperatures. Both processes usually produce thin piezoresistors, typically 100s of nm thick in structures that are microns or tens of microns thick. These aspect ratios are challenging to resolve with a practical mesh, so the MEMS Module includes physics interfaces that make it possible to model the current conduction in a thin layer, represented by a boundary in the model geometry. The Piezoresistivity, Boundary Currents interface is appropriate for situations in which the thickness of the piezoresistive layer is much less than that of the structural layer. If the structural layer is also sufficiently thin that it can be represented using the Shell interface, then the Piezoresistivity, Shell interface can be used (requires the Structural Mechanics Module

in addition to the MEMS Module). This circumstance is frequently encountered in modeling pressure sensors, which usually have piezoresistors embedded near the edge of large membranes. Note that the Piezoresistive Layer is still assumed to be much thinner than the structural layer in this physics interface. Finally if both the conducting and structural layers are meshed through their thicknesses the Piezoresistivity, Domain Currents interface can be used. This physics interface allows for advanced modeling of, for example, nonuniform doping profiles in the piezoresistor at the expense of additional degrees of freedom in the model.

- The Piezoresistivity, Domain Currents Interface
- The Piezoresistivity, Boundary Currents Interface
- The Piezoresistivity, Shell Interface
- Theory for the Piezoresistivity Interfaces

Thermal Effects in MEMS Devices

In this section:

- About Thermal Effects in MEMS Devices
- Modeling Thermal Actuators
- Modeling Thermal Stresses
- Thermoelastic Damping
- Physics Interfaces for Modeling Heat Transfer in Solids

> The Heat Transfer in Solids Interface in the *COMSOL Multiphysics Reference Manual*

About Thermal Effects in MEMS Devices

Heat transfer is characterized by three mechanisms: *conduction*, *convection*, and *radiation*. In MEMS actuators and sensors heat transfer by conduction is usually the dominant mechanism. Conductive heat transfer is diffusive and is driven by a temperature gradient. Within a solid the heat transfer equations take the following form:

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T) = Q \qquad (2\text{-}14)$$

Here ρ is the density of the solid, C_p is its heat capacity, k is Boltzmann's constant, T is the temperature, and Q is a term that includes heat sources and sinks. The Heat Transfer in Solids interface solves Equation 2-14, without additional coupling assumed.

Equation 2-14 is solved in the Joule Heating interface, along with the equations for electric current flow. In this case the volumetric heat source Q takes the value:

$$Q = \mathbf{J} \cdot \mathbf{E} \qquad (2\text{-}15)$$

where $\mathbf{J}$ is the current density, and $\mathbf{E}$ is the electric field. The electric field component in the direction of the current flow expresses the voltage drop per unit length of material in the direction of flow. Since the current density gives the current per unit area through a surface with normal parallel to the direction of flow, the quantity Q is

a power per unit volume, with dimensions W/m^3. The Joule Heating interface includes heat transfer and electromagnetic fields; in addition, solid mechanics is required to model thermal actuators. The MEMS Module includes The Joule Heating and Thermal Expansion Interface interface for this purpose.

Modeling Thermal Actuators

To model thermal actuators, The Joule Heating and Thermal Expansion Interface is required. This multiphysics interface adds a Solid Mechanics interface, a Heat Transfer in Solids interface, and an Electric Currents interface. Joule heating and the resulting thermal expansion are included as multiphysics coupling nodes under Multiphysics in the model tree. Note that a reference temperature, at which the thermal strains are zero, must be added to the Thermal Expansion node. In the absence of thermal residual stresses this temperature should be specified as the operating temperature.

> The Modeling Thermal Stresses section explains how to use this feature to include residual stresses in the film in addition to the thermal stresses generated by Joule heating.

Modeling Thermal Stresses

Some MEMS devices are fabricated by sequential depositions of thin or thick films. This process introduces thermal residual stresses in the structures due to the mismatched thermal expansivity of the deposited layers and the substrate as well as the difference between the deposition and operating temperatures.

Thermal stresses in a single added layer can be modeled in multiphysics interfaces using a coupled multiphysics interface that include the Solid Mechanics interface by adding a The Fluid-Solid Interaction Interface subnode under the Multiphysics node (in the Solid Mechanics interface you can also add it as subnode to the material model). A reference temperature, T_{ref}, for zero thermal strain must be specified, and the operating temperature, T, is set by selecting a heat transfer interface in a multiphysics model, which sets up this temperature coupling automatically, or by selecting a temperature as a model input (that is, temperatures generated by other physics interfaces can be selected or a user-defined value or expression for the temperature can be supplied), if the thermal expansion is defied in the Solid Mechanics interface. If the device is operated at a constant temperature, then The Solid Mechanics Interface should be used with T specified as a user-defined constant value in the model input. If

temperature gradients resulting from external heat sources are present in the structure then The Thermal Stress, Solid Interface is the natural choice because this physics interface couples heat transfer with the Solid Mechanics interface (in this case you only need to specify the strain reference temperature T_{ref} in the Thermal Expansion subnode; the temperature coupling to the heat transfer interface is set up by the Thermal Stress multiphysics interface). Finally for Joule-heated structures, The Joule Heating and Thermal Expansion Interface make it possible to use arbitrary zero strain reference temperatures, which enables the inclusion of thermal stress in addition to the Joule heating effects. For this physics interface you only need to specify the strain reference temperature T_{ref} in the Thermal Expansion subnode as in the Thermal Stress interface.

Thermal stresses in multilayered structures can also be modeled. As an example, consider a trilayer structure (a substrate plus two deposited layers), modeled using the Solid Mechanics interface with a Thermal Expansion subnode. For the first added layer, the zero strain temperature should be set to the deposition temperature of this layer, but the solution temperature (set through the model input), should be set to the deposition temperature of the second layer. A second study step should then be added with the output of the first study step supplied as an initial stress (this requires two Solid Mechanics interfaces to be added to the model, and some adjustment of the solver settings). For the second study step the zero strain reference temperature is chosen as the deposition temperature of the second added layer and the solution (model input) temperature should be set to the operating temperature. This process can naturally be extended to any number of deposited layers.

	Thermal Stresses in a Layered Plate: Application Library path **MEMS_Module/Actuators/layered_plate**

Thermoelastic Damping

When the product of the coefficient of thermal expansion and the temperature of the structure at rest is large, mechanical vibrations cause significant oscillations of the temperature field. The resulting energy transfer from the mechanical domain to the thermal domain is called *thermoelastic damping*. The Thermoelasticity Interface provides a predefined multiphysics interface to model thermoelastic damping.

Physics Interfaces for Modeling Heat Transfer in Solids

Table 2-2 gives a summary of the thermal physics interfaces showing which physics nodes are available in each interface.

TABLE 2-2: SUMMARY OF THE PHYSICS INTERFACES FOR MODELING HEAT TRANSFER AND THERMAL EXPANSION IN THE MEMS MODULE

PHYSICS INTERFACE	ID	HEAT TRANSFER INCLUDED	SOLID MECHANICS INCLUDED?	ELECTRIC CURRENTS AND JOULE HEATING INCLUDED?	MODELING THERMAL STRESSES
Heat Transfer in Solids	ht	Yes	No	No	Not applicable — would need to add **Solid Mechanics** separately.
Solid Mechanics	solid	No	Yes	No	Add **Thermal Expansion** node & specify T_{ref} and T.
Thermal Stress	—	Yes	Yes	No	Specify T_{ref} in the **Thermal Expansion** node under **Multiphysics**.
Joule Heating	—	Yes	No	Yes	Not applicable — would need to add **Solid Mechanics** separately.
Joule Heating and Thermal Expansion	—	Yes	Yes	Yes	Specify T_{ref} in the **Thermal Expansion** node under **Multiphysics**.

Microfluidics

COMSOL provides a dedicated Microfluidics Module that is designed for users developing microfluidic devices. This module includes features for modeling single-phase and multiphase flow, electrokinetics, flow in porous media, rarefied gas flows, and chemical transport.

In the MEMS Module, the microfluidics physics interfaces are focused on modeling the interaction of fluids with microstructures.

In this section:

- Fluid-Structure Interaction
- Thin-Film Damping

Fluid-Structure Interaction

The Fluid-Structure Interaction interface can be used to compute the flow of a liquid or gas around a moving microstructure. A typical application is the simulation of microvalves. The physics interface can be used to model both turbulent and laminar flows at low Mach numbers.

The Fluid-Structure Interaction interface uses the arbitrary Lagrangian-Eulerian (ALE) method to combine the fluid flow equations, formulated using an Eulerian description and a spatial frame, with the solid mechanics equations, formulated using a Lagrangian description and a material (reference) frame. The Fluid-Structure Interaction couplings appear on the boundaries between the fluid and the solid.

- Damping and Losses
- The Fluid-Solid Interaction Interface (in the *Structural Mechanics Module User's Guide*)
- The Piezoelectricity Interface, Solid (in the *Structural Mechanics Module User's Guide*)

Thin-Film Damping

In many MEMS devices a released structure moves relative to the substrate, compressing or shearing a thin layer of gas between the two structures. The fluid film

can be the dominant form of damping for the moving microstructure. If the film is compressed by a structure moving perpendicular to its plane, this type of damping is usually referred to as *squeeze-film damping*. Damping induced by motion of the structure parallel to the plane in which the fluid flows is called *slide-film damping*.
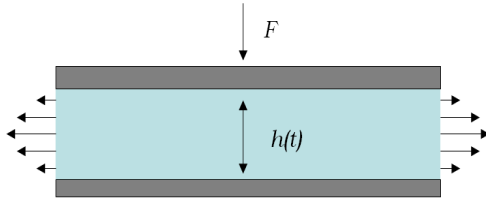
Figure 2-2: Diagram showing a circumstance in which squeeze-film damping applies.

It is often convenient to model the film using the Reynolds equation — enabling the 3D gas layer to be modeled on a 2D surface. The Reynolds equation applies for isothermal flows of a thin film in which the film height, h, is much less than a typical lateral dimension of the device, L. It must also be possible to neglect inertial flow effects in comparison to the viscous effects (for MEMS devices, this assumption is reasonable below MHz frequencies). Additionally, the curvature of the reference surface is neglected.

For isothermal, ideal gases the Reynolds equation can be simplified, leading to a modified Reynolds equation, formulated in the pressure only (the full Reynolds equation is formulated using both the pressure and the density). COMSOL Multiphysics includes both forms of the Reynolds equations in the Thin-Film Flow interfaces, and it is possible to switch between them using the Equation Type setting. Both forms of the equations include squeeze-film damping and slide-film damping.

The Thin-Film Flow interfaces are available as standalone physics interfaces that can be manually coupled to, for example, the physics interfaces in the Structural Mechanics branch. Thin-Film Damping has also been added as a boundary condition to the structural mechanics physics interfaces — these include Solid Mechanics, Piezoelectricity, Electromechanics, and Joule Heating and Thermal Expansion. The boundary condition automatically couples the damping forces to the model.

For gas damping, especially in vacuum packaged devices, the Knudsen number for the thin film gap is an important parameter to consider. This is the ratio of the gas mean free path to the gap height. Below Knudsen numbers of 0.01 it is necessary to use a slip boundary condition for the fluid by specifying a characteristic slip length. As the Knudsen number decreases further additional rarefaction effects need to be accounted

for. These can be included empirically in the Reynolds equation using models that require the mean free path and the tangential momentum accommodation coefficient (TMAC) to be specified. The TMAC is a measure of the mean fractional loss of tangential momentum of the gas molecules on collision with the walls — typical values for various gas surface combinations are available in Ref. 1 (see Table 6 in this reference).

	In the *CFD Module User's Guide*:
Q	• The Thin-Film Flow Interfaces
	• Theory for the Thin-Film Flow Interfaces

Reference for Thin-Film Damping

1. F. Sharipov, "Data on the Velocity Slip and Temperature Jump on a Gas-Solid Interface", *J. Phys. Chem. Ref. Data*, vol. 40, no. 2, 023101, 2011.

Connecting to Electrical Circuits

In this section:

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

About Connecting Electrical Circuits to Physics Interfaces

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

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

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

These features either accept a voltage measurement from the connecting noncircuit physics interface and return a current from an Electrical Circuit interface or the other way around.

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

Connecting Electrical Circuits Using Predefined Couplings

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

This coupling is typically activated when:

- A choice is made in the **Settings** window for the noncircuit physics interface feature, which then announces (that is, includes) the coupling to the Electrical Circuit interface. Its voltage or current is then included to make it visible to the connecting circuit feature.

- A voltage or current that has been announced (that is, included) is selected in a feature node's **Settings** window.

These circuit connections are supported in Terminals.

Connecting Electrical Circuits by User-Defined Couplings

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

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

- Define your own voltage or current measurement in the noncircuit physics interface using variables, coupling operators and so forth.

- In the **Settings** window for the Electrical Circuit interface feature, selecting the User-defined option and entering the name of the variable or expression using coupling operators defined in the previous step.

DETERMINING A CURRENT OR VOLTAGE VARIABLE NAME

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

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

2 Expand the **Solver>Dependent Variables** node and click the state node, in this example, **Current through device R1 (comp1.currents)**. The variable name is shown in the **Settings** window for **State**.
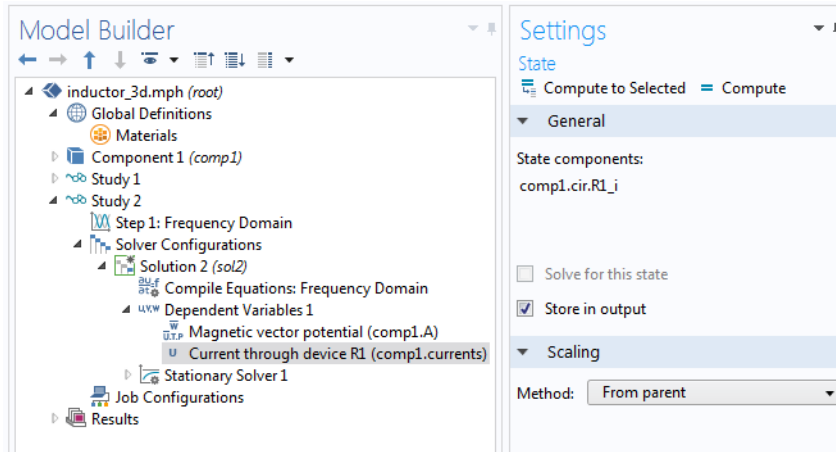
> 💡 Typically, voltage variables are named `cir.Xn_v` and current variables `cir.Xn_i`, where *n* is the "External" device number — 1, 2, and so on.

Solving

> ❗ Some modeling errors lead to the error message *The DAE is structurally inconsistent* being displayed when solving. This error typically occurs from having an open current loop, from connecting voltage sources in parallel, or connecting current sources in series.
>
> In this respect, the predefined coupling features are also treated as (ideal) voltage or current sources. The remedy is to close current loops and to connect resistors in series with voltage sources or in parallel with current sources.

Postprocessing

The Electrical Circuits interface, unlike most of the other physics interfaces, solves for a relatively large number of global dependent variables (such as voltages and currents),

instead of solving for a few space-varying fields (such as temperature or displacement). For this reason, the Electrical Circuit interface does not provide default plots when computing a study.

The physics interface defines a number of variables that can be used in postprocessing. All variables defined by the Electrical Circuit interface are of a global scope, and can be evaluated in a **Global Evaluation** node (under **Derived Values**). In addition, the time evolution or dependency on a parameter can be plotted in a **Global** plot (under a **1D Plot Group** node).

The physics interface defines a Node voltage variable for each electrical node in the circuit, with name `cir.v_name`, where `cir` is the physics interface Label and `<name>` is the node Name. For each two-pin component, the physics interface also defines variables containing the voltage across it and the current flowing through it.

🔍	In the *COMSOL Multiphysics Reference Manual:* • Derived Values, Evaluation Groups, and Tables and Global Evaluation • Plot Groups and Plots and Global

References

1. V. Kaajakari, *Practical MEMS*, Small Gear Publishing, Las Vegas, 2009.

2. S.D. Senturia, *Microsystem Design*, Springer Science and Business Media, New York, 2001.

3. A.F. Bower, *Applied Mechanics of Solids*, CRC Press, Boca Raton, FL, 2010 (http://www.solidmechanics.org).

SPICE Import and Export

The following text is a generic description of the SPICE Import and Export functionality available with various add-on products from COMSOL.

SPICE Import

The circuit definition in COMSOL Multiphysics adheres to the SPICE format developed at the University of California, Berkeley (Ref. 1). SPICE netlists can be imported and the corresponding circuit nodes are generated in the COMSOL Multiphysics model. Most circuit simulators can export to this format or some version of it.

The Electrical Circuit interface supports the following device models:

TABLE 2-3: SUPPORTED SPICE DEVICE MODELS

STATEMENT	DEVICE MODEL
R	Resistor
C	Capacitor
L	Inductor
V	Voltage Source
I	Current Source
E	Voltage-Controlled Voltage Source
F	Current-Controlled Current Source
G	Voltage-Controlled Current Source
H	Current-Controlled Voltage Source
D	Diode
Q	NPN BJT and PNP BJT
M	n-Channel MOSFET and p-Channel MOSFET
X	Subcircuit Instance

Statements corresponding to multiple devices are resolved by parsing the associated `.model` statement. The physics interface also supports the `.subckt` statement, which is represented in COMSOL by a Subcircuit Definition node, and the `.include` statement. SPICE commands are interpreted case-insensitively. The statement defining each device is also interpreted as the **Device name**.

According to SPICE specification, the first line in the netlist file is assumed to be the title of the netlist and it is ignored by the parser.

SPICE Export

The **SPICE Export** functionality creates a SPICE netlist file containing a description of the circuit represented by the physics interface. This functionality can be accessed from the physics interface context menu (right-click the physics interface node and select **Export SPICE Netlist**). After specifying a filename, the circuit is exported and messages from the export process display in the **Messages** window. During the export process, a series of operations are performed:

- In order to avoid conflicts, each component must be identified by a unique **Device name**. If one or more components have the same device name, the export operation fails and an error message is displayed. All characters in a **Device name** that are not letters, digits or underscores are replaced by underscores.

- According to the SPICE specification, each circuit must have a node with name 0, which is assumed to be the only ground node. When exporting a circuit, any node with name 0 that is not connected to a **Ground** component is exported with a different node name. All nodes that are connected to a Ground components are exported as a merged node with name 0. The Messages window shows a log message if these operations are performed, showing the name of the renamed or merged nodes.

- All characters in node names that are not letters, digits or underscores are replaced by underscores.

- Some components (most notably, the **External** components used to couple to other physics interfaces) cannot be exported to a SPICE netlist. These components are ignored during the export process, and a message is shown in the Messages window. Note that this can change the exported circuit, since some components are then missing.

- Subcircuit definitions are added as `.subckt` statements in the netlist. Semiconductor devices (such as MOSFETs, BJTs, and diodes) are exported as a SPICE device with a corresponding `.model` statement.

The title of the exported netlist file is the model's filename, and the time, date, and version of COMSOL Multiphysics is added as a comment in the netlist file.

Reference

1. http://bwrc.eecs.berkeley.edu/Classes/IcBook/SPICE/

3

AC/DC Interfaces

This chapter describes the physics interfaces found under the **AC/DC** branch ⚡.

In this chapter:

- The Electrostatics Interface
- The Electrostatics, Boundary Elements Interface
- The Electric Currents Interface
- The Electrical Circuit Interface
- Theory for the Electrostatics Interface
- Theory for the Electric Currents Interface
- Theory for the Electrical Circuit Interface

See the *COMSOL Multiphysics Reference Manual* for details about The Magnetic Fields Interface and the Theory of Magnetic Fields.

The Electrostatics Interface

The **Electrostatics (es)** interface (≽), found under the **AC/DC>Electric Fields and Currents** branch when adding a physics interface, is used to compute the electric field, electric displacement field, and potential distributions in dielectrics under conditions where the electric charge distribution is explicitly prescribed. The formulation is stationary except for when it is used together with other physics interfaces. Eigenfrequency, frequency-domain, small-signal analysis, and time-domain modeling are supported in all space dimensions.

The physics interface solves Gauss' law for the electric field using the scalar electric potential as the dependent variable.

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

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Charge Conservation**, **Zero Charge** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and space charges. You can also right-click **Electrostatics** to select physics features from the context menu.

Physics-Controlled Mesh
The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the **Sequence type** is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the check box in the **Use** column on the same table row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

Information from the physics, such as the presence of an infinite elements domain or periodic condition, will be used to automatically set up an appropriate meshing sequence.

| ! | In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh. |

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is es.

CROSS-SECTION AREA (1D COMPONENTS)

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

THICKNESS (2D COMPONENTS)

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

MANUAL TERMINAL SWEEP SETTINGS

Enter a **Reference impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω.

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

The generated lumped parameters are in the form of capacitance matrix elements. The terminal settings must consistently be of either fixed voltage or fixed charge type.

The lumped parameters are subject to **Touchstone file export**. Enter a file path or **Browse** for a file. Select a **Parameter format (value pairs)** for the Touchstone export — **Magnitude and angle (MA)** (the default), **Magnitude in dB and angle (DB)**, or **Real and imaginary parts (RI)**. Select an option from the **If file exists** list — **Overwrite** or **Create new**. Create new is useful when the model is solved multiple times with different settings. Select a **Parameter to export**: **Z** (the default), **Y**, or **S**.

From the **Home** toolbar add a **Parameters** node and enter the chosen name and assign to it a temporary **Expression** of unity into the **Parameters** table. You can also right-click **Global Definitions** to add the node.

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

	For most applications it is simpler to sweep over terminals using the Stationary Source Sweep study step. See Stationary Source Sweep in the *COMSOL Multiphysics Reference Manual* and (for more details) Stationary Source Sweep in the *AC/DC Module User's Guide*.

DEPENDENT VARIABLES

The dependent variable is the **Electric potential** V. You can change its name, which changes both the field name and the variable name. If the new name coincides with the name of another electric potential field in the model, the physics interfaces shares degrees of freedom. The new name must not coincide with the name of a field of another type or with a component name belonging to some other field.

DISCRETIZATION

Select the shape order for the **Electric potential** dependent variable — **Linear**, **Quadratic** (the default), **Cubic**, **Quartic**, or **Quintic**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

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

🔍	• Domain, Boundary, Edge, Point, and Pair Nodes for the Electrostatics Interface • Theory for the Electrostatics Interface

▥	*Electric Sensor*: Application Library path **COMSOL_Multiphysics/ Electromagnetics/electric_sensor**

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

The Electrostatics interface has these domain, boundary, edge, point, and pair nodes available.

ABOUT THE BOUNDARY CONDITIONS

The relevant physics interface condition at interfaces between different media is

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

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

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

AVAILABLE NODES

These nodes, listed in alphabetical order, are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users). Also see Table 3-1 for a list of interior and exterior boundary conditions, including edge, point, and pair availability.

💡	In general, to add a node, go to the **Physics** toolbar no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Change Cross Section
- Change Thickness (Out-of-Plane)
- Charge Conservation
- Charge Conservation, Piezoelectric[1,2]
- Charge Conservation, Ferroelectric[2,3]
- Conduction Loss (Time-Harmonic)
- Dispersion
- Dielectric Shielding
- Distributed Capacitance
- Electric Displacement Field
- Electric Potential
- Electrostatic Point Dipole
- External Surface Charge Accumulation
- Floating Potential
- Force Calculation
- Ground
- Initial Values
- Line Charge
- Line Charge (on Axis)
- Line Charge (Out-of-Plane)
- Periodic Condition
- Point Charge
- Point Charge (on Axis)
- Space Charge Density
- Surface Charge Density
- Symmetry Plane (for Electric Field)
- Terminal
- Thin Low Permittivity Gap
- Zero Charge (the default boundary condition)

[1]This feature is available with the Piezoelectricity multiphysics interface.

[2]Requires either the Acoustics Module, MEMS Module, or Structural Mechanics Module.

[3]This feature is available with the Ferroelectroelasticity multiphysics interface.

Table 3-1 lists the interior and exterior boundary conditions available with this physics interface. It also includes edge, point, and pair availability.

TABLE 3-1: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS (INCLUDING EDGE, POINT, AND PAIR AVAILABILITY) FOR THE ELECTROSTATICS INTERFACE

NODE	INTERIOR	EXTERIOR	ALSO AVAILABLE FOR
Change Cross Section	x	x	pairs
Change Thickness (Out-of-Plane)	x	x	pairs
Dielectric Shielding	x	x	pairs
Distributed Capacitance		x	pairs
Electric Displacement Field	x	x	pairs
Electric Potential	x	x	edges, points, and pairs

TABLE 3-1: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS (INCLUDING EDGE, POINT, AND PAIR AVAILABILITY) FOR THE ELECTROSTATICS INTERFACE

NODE	INTERIOR	EXTERIOR	ALSO AVAILABLE FOR
External Surface Charge Accumulation		x	pairs
Floating Potential	x	x	pairs
Ground	x	x	edges, points, and pairs
Periodic Condition		x	not applicable
Surface Charge Density	x	x	pairs
Symmetry Plane		x	not applicable
Terminal	x	x	domains
Thin Low Permittivity Gap	x		not applicable
Zero Charge (the default)	x	x	pairs

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

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

Charge Conservation

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

MATERIAL TYPE

The **Material type** setting decides how materials behave and how material properties are interpreted when the mesh is deformed. Select **Solid** for materials whose properties change as functions of material strain, material orientation, and other variables evaluated in a material reference configuration (material frame). Select **Nonsolid** for

materials whose properties are defined only as functions of the current local state at each point in the spatial frame, and for which no unique material reference configuration can be defined. Select **From material** to pick up the corresponding setting from the domain material on each domain.

CONSTITUTIVE RELATION D-E

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

- **Relative permittivity** (the default) to use the constitutive relation $\mathbf{D} = \varepsilon_0\varepsilon_r\mathbf{E}$. Then the default is to take the **Relative permittivity** ε_r (dimensionless) values **From material**. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix. The default is 1.

- **Polarization** to use the constitutive relation $\mathbf{D} = \varepsilon_0\mathbf{E} + \mathbf{P}$. Then enter the components based on space dimension for the **Polarization** vector $\mathbf{P}$ (SI unit: C/m^2). The defaults are 0 C/m^2.

- **Remanent electric displacement** to use constitutive relation $\mathbf{D} = \varepsilon_0\varepsilon_r\mathbf{E} + \mathbf{D}_r$, where $\mathbf{D}_r$ is the remanent displacement (the displacement when no electric field is present). Then the default is to take the **Relative permittivity** ε_r (dimensionless) values **From material**. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix. Then enter the components based on space dimension for the **Remanent electric displacement** $\mathbf{D}_r$ (SI unit: C/m^2). The defaults are 0 C/m^2.

- **Dispersion** to use the constitutive relation $\mathbf{D} = \varepsilon_0\mathbf{E} + \mathbf{P}(\mathbf{E}, \varepsilon_{rS})$, where the polarization vector is calculated from the electric field using the dielectric dispersion model. This option is available when the **Material type** is set to **Solid**. You enter the **Relative permittivity** ε_{rS} (dimensionless) values **From material**. For **User defined**, select **Isotropic**, **Diagonal**, or **Symmetric** and enter values or expressions in the field or matrix. This value of electric permittivity will be used in stationary study, for which the polarization vector is calculated as $\mathbf{P} = \varepsilon_0(\varepsilon_{rS} - I)\mathbf{E}.$

	Once this option is selected, a subnode Dispersion will become available under the **Charge Conservation** node. At that subnode, you can select the dispersion model, enter the corresponding parameters, and chose how the relative permittivity input on the parent node will be interpreted in Eigenfrequency, Frequency Domain and Time Dependent studies.

Conduction Loss (Time-Harmonic)

	This feature requires either the Acoustics Module, or MEMS Module, or Structural Mechanics Module. See the individual documentation for information.

The **Conduction Loss (Time-Harmonic)** subnode to **Charge Conservation** allows you to model possible conductive losses in a dielectric material. The effect is only active in an Eigenfrequency or Frequency Domain study.

CONDUCTION CURRENT

By default, the **Electrical conductivity** σ for the media is defined **From material**. You can also select **User defined** or **Linearized resistivity**.

- For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** depending on the characteristics of the electrical conductivity, and then enter values or expressions for the **Electrical conductivity** σ in the field or matrix.

- For **Linearized resistivity** the default **Reference temperature** T_{ref}, and **Resistivity temperature coefficient** α, and **Reference resistivity** ρ_0 are taken **From material**, which means that the values are taken from the domain (or boundary) material. T is the current temperature, which can be a value that is specified as a model input or the temperature from a heat transfer interface. The definition of the temperature field appears in the **Model Inputs** section.

LOCATION IN USER INTERFACE

Context Menus
Electrostatics>Charge Conservation>Conduction Loss (Time-Harmonic)

Ribbon
Physics tab with **Charge Conservation** node selected in the model tree:

Attributes>Conduction Loss (Time-Harmonic)

Initial Values

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

INITIAL VALUES

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

Space Charge Density

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

SPACE CHARGE DENSITY

Enter a value or expression for the **Space charge density** ρ_v (SI unit: C/m^3). The default is 0 C/m^3.

Force Calculation

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

FORCE CALCULATION

Enter a **Force name**, which is then appended to global variables. The method used to compute forces and torques is integration of the Maxwell's stress tensor over the exterior surfaces of the set of domains. This feature also gives access to the normal component of the Maxwell stress tensor on the external surfaces. The computed force is made available as a global vector variable with name
`<name>.Force<component>_<force name>`, where `<name>` is the **Name** of the physics interface.

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

In frequency domain studies, the cycle-averaged force and torque variables are also defined, with names `<name>.Forceav<component>_<force name>`,

`<name>.Tav<component>_<force name>` and `<name>.Taxav_<force name>`. The variables described are still defined and represent the instantaneous force and torque.

⚠️
- In 2D axisymmetry, the radial component of the total force is set to zero, as this is an intrinsic property of the used symmetry.
- In small-signal analysis studies, no cycle-average variables are defined. Use the additional plot settings for the Harmonic Perturbation on the instantaneous variables to compute the quantities of interest. See Harmonic Perturbation, Prestressed Analysis, and Small-Signal Analysis in the *COMSOL Multiphysics Reference Manual*.

🔍 For more information on calculation of forces and torques, see Electrostatics.

ADVANCED SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

The **Compute both instantaneous and cycle-averaged variables in frequency domain** check box is selected by default. The check box controls which variables are defined by the Force Calculation feature in frequency-domain studies. When selected (the default setting in new models), the variables are defined as described above. This check box selected by default for backward compatibility. If the check box is cleared, only the cycle-averaged variables are computed, replicating the behavior of the Force Calculation feature in version 4.3 and earlier. The check box is cleared in existing Force Calculation features when opening models created in version 4.3 and earlier.

Zero Charge

The **Zero Charge** node adds the condition that there is zero charge on the boundary so that $\mathbf{n} \cdot \mathbf{D} = 0$. This boundary condition is also applicable at symmetry boundaries where the potential is known to be symmetric with respect to the boundary. This is the default boundary condition at exterior boundaries. At interior boundaries, it means that no displacement field can penetrate the boundary and that the electric potential is discontinuous across the boundary.

Ground

The **Ground** node implements ground (zero potential) as the boundary condition $V = 0$.

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

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

BOUNDARY, EDGE, OR POINT SELECTION

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

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

Electric Potential

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

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

BOUNDARY, EDGE, OR POINT SELECTION

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

Enter the value or expression for the **Electric potential** V_0 (SI unit: V). The default is 0 V.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

Surface Charge Density

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

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

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

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or click **Harmonic Perturbation** on the **Physics** toolbar. For more information, see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

SURFACE CHARGE DENSITY

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

External Surface Charge Accumulation

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

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

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

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

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

MATERIAL TYPE

The **Material type** setting decides how materials behave and how material properties are interpreted when the mesh is deformed. Select **Solid** for materials whose properties change as functions of material strain, material orientation, and other variables evaluated in a material reference configuration (material frame). Select **Nonsolid** for materials whose properties are defined only as functions of the current local state at each point in the spatial frame, and for which no unique material reference configuration can be defined. Select **From material** to pick up the corresponding setting from the domain material on each domain.

EXTERNAL SURFACE CHARGE ACCUMULATION

Enter values or expressions for the **Normal ion current density** $\mathbf{n} \cdot \mathbf{J}_i$ (SI unit: A/m^2) sand the **Normal electron current density** $\mathbf{n} \cdot \mathbf{J}_e$ (SI unit: A/m^2).

Symmetry Plane (for Electric Field)

The **Symmetry Plane** node adds a boundary condition that represents symmetry or antisymmetry in the electric field, depending on which option is chosen.

SYMMETRY TYPE

Choose between **Symmetry** and **Antisymmetry** for the electric field. If **Symmetry** is chosen, the boundary condition is

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

which states that the normal component of the electric field is zero. If **Antisymmetry** is chosen, the boundary condition becomes

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

which states that the tangential component of the electric field is zero.

Electric Displacement Field

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

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

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

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

Periodic Condition

The **Periodic Condition** node defines periodicity or antiperiodicity between two boundaries. If required, activate periodic conditions on more than two boundaries, in which case the Periodic Condition tries to identify two separate surfaces that can each consist of several connected boundaries. For more complex geometries, it might be necessary to use the **Destination Selection** section, which you activate by right-clicking the **Periodic Condition** node and select **Manual Destination Selection**. With that section, the boundaries which constitute the source and destination surfaces can be manually specified.

📄	When this feature is used in conjunction with a Sector Symmetry feature on connected boundaries, wherever the sector symmetry boundaries connect with the periodic boundaries, the same periodic condition feature cannot be used on both sides. At least two periodic condition features are required for the model to compute correctly.

BOUNDARY SELECTION

📄	The software usually automatically identifies the boundaries as either source boundaries or destination boundaries, as indicated in the selection list. This works fine for cases like opposing parallel boundaries. In other cases, right-click **Periodic Condition** and select **Manual Destination Selection** to control the destination. By default it contains the selection that COMSOL Multiphysics identifies.

When using nonconforming meshes on the source and destination of a periodic boundary pair, for numerical stability, a finer mesh should be applied on the destination side. Use conforming meshes if possible.

DESTINATION SELECTION

This section is available for specifying the destination boundaries, if needed, when the **Manual Destination Selection** option is selected in the context menu for the **Periodic Condition** node. You can only select destination boundaries from the union of all source and destination boundaries.

PERIODIC CONDITION

Select a **Type of periodicity** — **Continuity** (the default), **Antiperiodicity**, or **Floquet periodicity**. Select:

- **Continuity** to make the potential periodic (equal on the source and destination).

- **Antiperiodicity** to make it antiperiodic.

- **Floquet periodicity** (only available with products supporting piezoelectric modeling). Specify the components of the **k-vector for Floquet periodicity** $\mathbf{k}_F$ (SI unit: rad/m).

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog box.

ORIENTATION OF SOURCE

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog box. For information about the **Orientation of Source** section, see Orientation of Source and Destination.

ORIENTATION OF DESTINATION

This section appears if the setting for **Transform to intermediate map** in the **Orientation of Source** section is changed from the default value, **Automatic**, and **Advanced Physics Options** is selected in the **Show More Options** dialog box. For information about the **Orientation of Destination** section, see Orientation of Source and Destination.

Thin Low Permittivity Gap

Use the **Thin Low Permittivity Gap** node

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

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

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

THIN LOW PERMITTIVITY GAP

The default is to take the **Relative permittivity** ε_r (dimensionless) values **From material**. For **User defined**, enter a different value or expression. Enter a **Thickness** d (SI unit: m). The default is 5 mm.

Dielectric Shielding

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

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

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

MATERIAL TYPE

The **Material type** setting decides how materials behave and how material properties are interpreted when the mesh is deformed. Select **Solid** for materials whose properties change as functions of material strain, material orientation and other variables evaluated in a material reference configuration (material frame). Select **Nonsolid** for materials whose properties are defined only as functions of the current local state at each point in the spatial frame, and for which no unique material reference configuration can be defined. Select **From material** to pick up the corresponding setting from the domain material on each domain.

ELECTRIC SHIELDING

The default is to take the **Relative permittivity** ε_r (dimensionless) values **From material**. It takes it from the adjacent domains if not explicitly defined. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix. Enter a **Surface thickness** d_s (SI unit: m) of the shielding. The default is 1 m.

Terminal

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

An operation called **Split by Connectivity** ( ) is available by right-clicking the parent node. The operation is useful when there are several terminal domains or boundaries

that are not geometrically connected. Clicking the operation node will automatically generate several **Terminal** nodes where each selection is geometrically connected.

The **Terminal** node also enables the computation of the lumped parameters of the system, such as capacitance.

TERMINAL

Specify the terminal properties. Enter a unique **Terminal name** that will be used to identify the global variables (such as the terminal voltage or current) that are defined by the terminal. The **Terminal name** can contain alphanumeric characters, but it should be numeric when performing terminal sweeps.

Select a **Terminal type** — **Charge** (the default), **Voltage**, **Circuit**, or **Terminated**. Select:

- **Charge** to prescribe the total charge deposited on the terminal boundaries. Enter the total charge Q_0. The default is 0 C, which corresponds to a uncharged floating electrode.

- **Voltage** to specify a fixed electric potential at the terminal. Enter an electric potential V_0. The default is 1 V.

- **Circuit** to specify a terminal connected to an Electrical Circuit physics interface that models an external circuit. The Terminal node provides a current-voltage characteristic to the circuit.

- **Terminated** to connect the terminal to an impedance that might represent a load or a transmission line. When **Terminated** is selected, the scattering parameters (S-parameters) are computed. The **Terminated** setting can only be used in frequency-domain studies.

  - Enter a **Terminal power** P_0 to specify the input power at this terminal from the transmission line. This excitation can be toggled using a port sweep.

  - Select an option from the **Characteristic impedance from** list to define the value of the impedance — **Physics interface** or **User defined**. For **Physics interface**, the **Reference impedance** Z_{ref} defined on the **Settings** window for the physics under Sweep Settings is used. For **User defined**, enter a specific impedance Z_{ref} for this terminal. The default is 50 Ω.

When **Terminal type** is selected as **Charge** or **Voltage**, a contribution can be added as **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** on the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

Initial values

The initial values subsection is used to provide initial values for the internal state variables in the Terminal. These settings can be useful to provide an initial guess to the nonlinear solver or the initial value in a time-dependent study.

The **Initial value for voltage** is available for all Terminal types except Voltage. Enter an **Initial value for voltage** V_{init}.

The **Initial value for charge** is available for the Circuit excitation. It specifies the initial charge on the terminal when the circuit is connected. Enter an **Initial value for charge** Q_{init}.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

Floating Potential

The **Floating Potential** node is used when modeling a metallic electrode at floating potential. It applies a constant voltage V_0 on the boundary (for domain features, this is the boundary enclosing the selected domain), such that the total normal electric displacement field **D** equals a specific charge Q_0:

$$V = V_0$$

$$\int_{\partial\Omega} (\mathbf{D} \cdot \mathbf{n})dS = Q_0,$$

where $\partial\Omega$ represents the boundary and **n** refers to the surface normal. The constant boundary voltage implies the tangential electric field equals zero; the electric field will be perpendicular to the boundary:

$$-\nabla_t V = \mathbf{E}_t = 0$$
$$\mathbf{n} \times \mathbf{E} = 0.$$

In case of $Q_0 = 0$ (the default case), the boundary will behave as an unconnected, neutrally charged, good conductor under electrostatic conditions (a floating equipotential). Although locally the displacement field may vary, the total electric displacement field entering or leaving the boundary equals zero.

If the floating potential touches a point, boundary or domain feature that is not floating (a **Terminal** or **Ground** feature), the floating potential will acquire that feature's

potential. If the floating potential is set to a certain charge, or connected to a circuit, it behaves like a terminal.

FLOATING POTENTIAL

The **Floating potential identifier** text area shows the unique identifier for the floating potential feature. It is used to identify the global variables created by the node, such as the voltage. The **Floating potential group** check box on the **Settings** window for **Floating Potential** controls how potentials are assigned to boundary segments in the feature's boundary selection. If this check box is not selected, a single potential is applied to all boundaries in the selection. If the check box is selected, each group of contiguous boundaries in the selection is given a unique potential. This simplifies the setup of models with many floating electrodes. The values of the potential at each group of boundaries are then made available in postprocessing, collected in a vector variable.

The following options are not available if the **Floating potential group** check box is selected. Select a specification for the **Electric charge** — choose **User defined** to specify a total **Charge** Q_0 deposited on the surface. Select **Circuit** to connect the floating potential to an Electrical Circuit.

Initial Values
The initial values subsection is used to provide initial values for the internal state variables in the Floating potential. These settings can be useful to provide an initial guess to the nonlinear solver or the initial value in a time-dependent study.

Enter an **Initial value for voltage** V_{init}.

The **Initial value for charge** is available for the Circuit excitation. It specifies the initial charge on the floating potential when the circuit is connected. Enter an **Initial value for charge** Q_{init}.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog box.

🔍	Settings and Properties Windows for Feature Nodes in the *COMSOL Multiphysics Reference Manual*

Distributed Capacitance

The **Distributed Capacitance** node adds a distributed capacitance boundary condition according to the following equations for exterior boundaries:

$$-\mathbf{n} \cdot \mathbf{D} = \varepsilon_0 \varepsilon_r \frac{V_{ref} - V}{d_s}$$

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

DISTRIBUTED CAPACITANCE

Enter the values or expressions for **Relative permittivity** ε_r (dimensionless), **Surface thickness** d_s (SI unit: m), and **Reference potential** V_{ref}. The default value for the surface thickness is 10^{-3} m (1 mm) and 0 V for the reference potential.

Line Charge

For 3D components, use the **Line Charge** node to specify line charges along the edges of a geometry. Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** on the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

EDGE SELECTION

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

LINE CHARGE

Enter a value or expression to apply a **Line charge** Q_L (SI unit: C/m). This source represents electric charge per unit length and the default is 0 C/m.

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

Line Charge (on Axis)

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

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or click **Harmonic Perturbation** on the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

LINE CHARGE (ON AXIS)

Enter a value or expression to apply a **Line charge** Q_L (SI unit: C/m). This source represents electric charge per unit length and the default is 0 C/m.

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

Line Charge (Out-of-Plane)

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

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** on the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

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

POINT SELECTION

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

LINE CHARGE (OUT-OF-PLANE)

Enter a value or expression to apply a **Line charge** Q_L (SI unit: C/m). This source represents electric charge per unit length and the default is 0 C/m.

🔍	Line Charge and Line Charge (on Axis)

Point Charge

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

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** on the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

POINT SELECTION

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

POINT CHARGE

Enter a value or expression to apply a **Point charge** Q_P (SI unit: C) to points. This source represents an electric displacement field flowing out of the point. The default is 0 C.

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

Point Charge (on Axis)

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

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** on the **Physics** toolbar. For more information see

Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

POINT SELECTION

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

POINT CHARGE (ON AXIS)

Enter a value or expression to apply a **Point charge** Q_P (SI unit: C) to points on an axis. This source represents an electric displacement field flowing out of the point. The default is 0 C.

🔍 | Point Charge and Line Charge (Out-of-Plane)

Change Cross Section

This node is available with 1D components. This setting overrides the global **Cross-Section Area** setting made in any physics interface that uses this feature. For 2D components, see Change Thickness (Out-of-Plane).

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

CHANGE CROSS SECTION

Enter a value or expression for the **Cross-section area** A. The default value of one unit area is typically not representative for the actual domain. Instead, it describes a unit area that makes the 1D equation identical to the equation used for 3D components.

Change Thickness (Out-of-Plane)

This node is available for 2D components. This setting overrides the global **Thickness** setting made in any physics interface that uses this node. For 1D components, see Change Cross Section.

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

CHANGE THICKNESS (OUT-OF-PLANE)

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

Charge Conservation, Ferroelectric

> This feature is available with the Ferroelectroelasticity multiphysics interface which requires either the Acoustics Module, MEMS Module, or Structural Mechanics Module. See Ferroelectroelasticity (in the *Structural Mechanics Module User's Guide*) for more information.

The **Charge Conservation, Ferroelectric** node is normally used together with a **Electrostriction** multiphysics coupling node and **Solid Mechanics** interface. The node is added by default to the **Electrostatics** interface when adding a Ferroelectroelasticity interface. It is also available from the context menu (right-click the **Electrostatics** interface parent node) or from the **Physics** toolbar.

Charge Conservation, Piezoelectric

> This feature is available with the Piezoelectricity interface, which requires either the Acoustics Module, or MEMS Module, or Structural Mechanics Module. See the individual documentation for information.

The **Charge Conservation, Piezoelectric** node is normally used together with a **Piezoelectric Effect** multiphysics coupling node and a corresponding **Piezoelectric Material** node in the **Solid Mechanics** interface. The node is added by default to the **Electrostatics** interface when adding a Piezoelectricity interface. It is also available from

the context menu (right-click the **Electrostatics** interface parent node) or from the **Physics** toolbar.

> ⚠ When the **Charge Conservation, Piezoelectric** node is added to the electrostatics interface in the absence of an active **Piezoelectric Effect** multiphysics coupling node, the material behaves similarly to a Charge Conservation node, with electric properties corresponding to the relative permittivity entered (see below). The piezoelectric effect is *not* included in the corresponding equation system.

ELECTRIC DISPLACEMENT

If the node is used together with an active **Piezoelectric Effect** multiphysics coupling node, then these settings are locked. Note that if they are unlocked, then the material behaves like a dielectric and *not* a piezoelectric. In this case, the default is to take the **Relative permittivity** ε_{rS} (dimensionless) values **From material**. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix.

Dispersion

The **Dispersion** subnode to **Charge Conservation** and **Charge Conservation, Piezoelectric** allows you to model possible dielectric losses in the material. The effect can be accounted for in Eigenfrequency, Frequency Domain and Time Dependent study.

Under the **Material model**, you can select either **Debye** (default) or **Multipole Debye** dispersion option.

For a single pole **Debye** dispersion choice, you can enter the **Relaxation time** and **Relative permittivity contribution**. In the Thermal Effects tab, you can select the **Shift function** type, which will be used to compute the effective relaxation time accounting for the temperature effects.

For the **Multipole Debye** dispersion, you can select two options to enter the **Relaxation Data**.

- **Relaxation time and Relative permittivity change** choice will provide an input table with dynamically changed size, where you can enter any number of **Relaxation time** and the corresponding **Relative permittivity contribution** values. In the Thermal Effects tab, you can select the **Shift function** type, which will be used to compute the effective relaxation times accounting for the temperature effects.

Constant Loss Tangent option. In this case, you enter the **Loss Tangent** $\eta(f_c)$ together with the **Center frequency** f_c. You also specify the model **Bandwidth (decades)** that defines a frequency interval centered at f_c, in which the loss tangent will be approximately constant and equal to $\eta(f_c)$. You can also select the **Accuracy** for the approximation to be either **Normal** (default) or **High**. The software will automatically deduce the necessary number of Debye poles together with the values of the corresponding relaxation times and relative permittivity contributions, which will be used in computations to maintain the requested bandwidth and accuracy.

⊗	Debye Dispersion

⚠	In Eigenfrequency study and Time Dependent study, each pole will require one extra vector degree of freedom (domain variable) to represent the corresponding contribution to the polarization. Thus, using many poles in the relaxation data input table, or setting the **Accuracy** to **High** accuracy in case of **Constant Loss Tangent**, can lead to significant computation costs for larger models.

For all cases, you can specify how the relative permittivity input, ε_{rS}, on the parent node should be interpreted by selecting the **Static response** (the default value is **High frequency limit**). This setting will have effect in Eigenfrequency, Frequency Domain, and Time Dependent study types.

THERMAL EFFECTS

Dispersion properties depend on the temperature. For many materials, a change in the temperature can be transformed directly into a change in the time scale. Thus, the relaxation time is modified to $a_T(T)\tau_m$, where $a_T(T)$ is a shift function.

Select a Shift function — **None**, **Vogel-Fulcher**, **Arrhenius**, **Williams-Landel-Ferry**, **Tool-Narayanaswamy-Moynihan**, or **User defined**.

- When the default, **None**, is kept, the shift function $a_T(T)$ is set to unity and the relaxation time is not modified.
- For **Vogel-Fulcher** enter values or expressions for these properties:
  - **Reference temperature** T_0. The default is 230 K.
  - **Activation energy** Q. The default is 8000 J/mol.

- For **Arrhenius** enter values or expressions for these properties:
  - **Reference temperature** T_0. The default is 293.15 K.
  - **Activation energy** Q.

- For **Williams-Landel-Ferry** enter values or expressions for these properties:
  - **Reference temperature** T_{WLF} The default is 293.15 K.
  - **WLF constant 1** $C_{1\mathrm{WLF}}$. The default is 17.44.
  - **WLF constant 2** $C_{2\mathrm{WLF}}$. The default is 51.6 K.

- For **Tool-Narayanaswamy-Moynihan** enter values or expressions for these properties:
  - **Reference temperature** T_0. The default is 293.15 K.
  - **Activation energy** Q.
  - **Activation energy fraction** χ $(0<\chi<1)$.
  - **Structural relaxation times** λ_0.
  - **Fictive temperature weights** w.

- For **User defined** enter a value or expression for the shift function a_{T}.

DISCRETIZATION

To display this section, click the **Show More Options** button (👁) and select **Discretization** from the **Show More Options** dialog box. Select the element order from the list box for the auxiliary electric field vector variables $\mathbf{e}_n$.

You can change the solution algorithm by using the check box **Use local time integration** (checked by default). When unchecked, the solution of the equations for the auxiliary dependent variables in Time Dependent study will be governed by the time stepper algorithm used in the solver.

LOCATION IN USER INTERFACE

Context Menus
Electrostatics>Charge Conservation, Piezoelectric>Dispersion

Ribbon
Physics tab with **Charge Conservation, Piezoelectric** node selected in the model tree:

Attributes>Dispersion

Electrostatic Point Dipole

Add an **Electrostatic Point Dipole** node to 3D and 2D components. Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or click **Harmonic Perturbation** on the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

The **Electrostatic Point Dipole** represents the limiting case of zero separation distance between two equally strong point sources of opposing signs while maintaining the product between separation distance and source strength at a fixed value (p). The dipole moment is a vector entity with positive direction from the negative charge to the positive one.

ELECTROSTATIC POINT DIPOLE

Select a **Dipole specification** — **Magnitude and direction** (the default) or **Electric dipole moment**.

- For **Magnitude and direction** enter coordinates for the **Electric dipole moment direction** $\mathbf{n}_p$ (dimensionless) and the **Electric dipole moment, magnitude** p.
- For **Electric dipole moment** enter coordinates for the **Electric dipole moment** $\mathbf{p}$.

The Electrostatics, Boundary Elements Interface

The **Electrostatics, Boundary Elements (esbe)** interface (⊞), found under the **AC/DC>Electric Fields and Currents** branch when adding a physics interface, is used for computing the potential distribution in dielectrics under conditions where the electric potential distribution on the boundaries is explicitly prescribed. The formulation is based on the boundary element method and the interface is available in 2D and 3D.

The physics interface solves Laplace's equation for the electric potential using the scalar electric potential as the dependent variable

$$-\nabla \cdot (\varepsilon_0 \varepsilon_r \nabla V) = 0 .$$

📄	For a discussion about the boundary element method, see Theory for the Boundary Elements PDE in the *COMSOL Multiphysics Reference Manual*.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Charge Conservation**, **Zero Charge** (on exterior boundaries and in 3D also on interior edges) and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, electric potential and surface charge density conditions. You can also right-click **Electrostatics, Boundary Elements** to select physics features from the context menu.

📄	If both The Electrostatics Interface and The Electrostatics, Boundary Elements Interface are available, the Electric Scalar-Scalar Potential Coupling node is available from the **Multiphysics** menu in the **Physics** toolbar or by right-clicking the **Multiphysics Couplings** node in **Model Builder**.
📄	The Electrostatics Interface and The Electrostatics, Boundary Elements Interface can also be coupled by using the same name for the dependent variable for both interfaces. Then Electric Scalar-Scalar Potential Coupling is not needed. How to set the name for the dependent variable is described in the Dependent Variables section.

Physics-Controlled Mesh

The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the **Sequence type** is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the check box in the **Use** column on the same table row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

Information from the physics will be used to automatically set up an appropriate mesh sequence.

> [!] In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh.

DOMAIN SELECTION

From the **Selection** list, select any of the options — **Manual**, **All domains**, **All voids**, or **All domains and voids** (the default). The geometric entity list displays the selected domain entity numbers. Edit the list of selected domain entity numbers using the selection toolbar buttons to the right of the list or by selecting the geometric entities in the **Graphics** window. Entity numbers for voids can be entered by clicking the Paste (📋) button in the selection toolbar and supplying the entity numbers in the dialog box. The entity number for the infinite void is 0, and finite voids have negative entity numbers.

Selections can also be entered using the **Selection List** window, available from the **Windows** menu on the **Home** toolbar.

> [📄] For more information about making selections, see Working with Geometric Entities in the *COMSOL Multiphysics Reference Manual*.

PHYSICS SYMBOLS

Select the **Enable physics symbols** check box to display symmetry planes (in 3D) and lines (in 2D) in the **Graphics** window, as specified in the Symmetry settings.

THICKNESS (2D COMPONENTS)

For 2D components, enter a default value for the **Out-of-plane thickness** d (SI unit: m). The default value of 1 is typically not representative for a thin dielectric medium.

Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D components.

MANUAL TERMINAL SWEEP SETTINGS

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

	For most applications it is simpler to sweep over terminals using the Stationary Source Sweep study step. See Stationary Source Sweep in the *COMSOL Multiphysics Reference Manual* and (for more details) Stationary Source Sweep in the Modeling with the AC/DC Module chapter of this manual.

	To correctly calculate the terminal charge and the corresponding capacitances, if voltage Terminal is used on boundaries between The Electrostatics Interface and The Electrostatics, Boundary Elements Interface, let the dependent variable for The Electrostatics, Boundary Elements Interface be the same as the dependent variable for The Electrostatics Interface and disable Electric Scalar-Scalar Potential Coupling. How to set the name for the dependent variable is described in the Dependent Variables section. Since the Electric Scalar-Scalar Potential Coupling is disabled, add manually a Fully Coupled node to the Stationary Solver below the Solution node in the Solver Configurations.

SYMMETRY

	For 3D components, from the **Condition for the $x = x_0$ plane**, **Condition for the $y = y_0$ plane**, and **Condition for the $z = z_0$ plane** lists, choose **Off** (the default), **Symmetric**, or **Antisymmetric**. Then enter the value for the plane location x_0, y_0, or z_0 (SI unit: m) as required.

For 2D components, from the **Condition for the x = x₀ plane** and **Condition for the y = y₀ plane** lists, choose **Off** (the default), **Symmetric**, or **Antisymmetric**. Then enter the value for the plane location x_0 or z_0 (SI unit: m) as required.

FAR-FIELD APPROXIMATION

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog box.

For more information about the **Far Field Approximation** settings, see Far-Field Approximation Settings in the *COMSOL Multiphysics Reference Manual*.

INFINITY CONDITION

For 3D components, select the **Infinity condition** — **Total charge** (the default) or **Asymptotic value at infinity**. Specify the **Total charge** Q_{tot} (SI unit: C; the default value is 0 C) or **Electric potential at infinity** V_∞ (SI unit: V; the default value is 0 V). If there is an antisymmetric symmetry in the potential field, it acts as an added infinite ground plane with a fixed value of the electric potential. The value at infinity is fixed to 0 by the presence of the infinite ground plane, so for this case there is a fixed **Zero potential at infinity** condition.

For 2D components, select the **Infinity condition** — **Total charge** (the default) or **Value at reference distance**. Specify the **Total charge** Q_{tot} (SI unit: C; the default value is 0 C) or **Electric potential at reference distance** V_{ref} (SI unit: V; the default value is 0 V). If there is an antisymmetric symmetry in the potential field, it acts as an added ground line with a fixed value of the electric potential. The value at a reference distance is fixed to 0 by the presence of the ground line, so for this case there is a fixed **Zero potential at reference distance** condition.

QUADRATURE

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

📄	For more information about the **Quadrature** settings, see Quadrature in the *COMSOL Multiphysics Reference Manual*.

DISCRETIZATION

From the **Electric potential/Surface charge density** list, choose from predefined options for the boundary element discretization order for the electric potential variable and the surface charge density variable, respectively. The predefined options represent the suitable combinations of element orders such as **Quadratic/Linear** (the default).

📦	For 3D components, the Line charge variable will have the same discretization order as the Electric potential variable.

The settings under **Value types when using splitting of complex variables** are important for sensitivity and optimization computations. See the description of the built-in operators fsens and fsensimag.

DEPENDENT VARIABLES

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

Domain, Boundary, and Edge Nodes for the Electrostatics, Boundary Elements Interface

The Electrostatics, Boundary Elements Interface has these domain, boundary, and edge nodes available from the **Physics** ribbon (Windows users), **Physics** context menu (Mac or Linux users), or by right-clicking the physics interface in **Model Builder** to access the context menu (all users).

💡	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

These nodes and subnodes are described in this section:

- Initial Values
- Charge Conservation
- Electric Potential
- Floating Potential
- Force Calculation

- Ground
- Surface Charge Density
- Terminal
- Zero Charge

The Zero Charge node is the default boundary and edge condition available for this physics interface and described for the Electrostatics interface.

Initial Values

Initial Values adds an initial value for the electric potential that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver. Add more **Initial Values** nodes from the **Physics** toolbar, **Global** menu.

INITIAL VALUES

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

Charge Conservation

The **Charge Conservation** node adds the equations for charge conservation according to Gauss' law for the electric displacement field. It provides an interface for defining the constitutive relation defined by the relative permittivity.

CONSTITUTIVE RELATION D-E

Enter the **Relative permittivity** ε_r (dimensionless) to describe the macroscopic properties of the medium. The default is to take the **Relative permittivity** values **From material**. For **User defined**, enter a value or expression in the field.

> **!** Notice that the boundary element method is based on the availability of an analytic Green's function for the domain. Thus, the **Relative permittivity** must evaluate to a constant value for each **Charge Conservation** node.

Electric Potential

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

!	For more information about the Electric Potential condition, see Electric Potential in the documentation for the Electrostatics interface.

	For 3D components, enter the **Edge radius** (SI unit: m) for the virtual tube surrounding the edge. The default value is 0.1 m. It is generally assumed that the edge radius is small in comparison to other geometrical measures of the model, such as the edge length and the distances between different edges in the geometry.

Floating Potential

The **Floating Potential** node is used when modeling a metallic electrode at floating potential.

!	For more information about the Floating Potential condition, see Floating Potential in the documentation for the Electrostatics interface.

	For 3D components, enter the **Edge radius** (SI unit: m) for the virtual tube surrounding the edge. The default value is 0.1 m. It is generally assumed that the edge radius is small in comparison to other geometrical measures of the model, such as the edge length and the distances between different edges in the geometry.

Ground

The **Ground** node implements ground (zero potential) as the boundary condition $V = 0$.

!	For more information about the Ground condition, see Ground in the documentation for the Electrostatics interface.

For 3D components, enter the **Edge radius** (SI unit: m) for the virtual tube surrounding the edge. The default value is 0.1 m. It is generally assumed that the edge radius is small in comparison to other geometrical measures of the model, such as the edge length and the distances between different edges in the geometry.

Surface Charge Density

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

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

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

For more information about the Surface Charge Density condition, see Surface Charge Density in the documentation for the Electrostatics interface.

For 3D components, enter the **Edge radius** (SI unit: m) for the virtual tube surrounding the edge. The default value is 0.1 m. It is generally assumed that the edge radius is small in comparison to other geometrical measures of the model, such as the edge length and the distances between different edges in the geometry. Enable **Compensate for tube volume** to allow for a potential gradient across the cross section of the virtual tube. This can be useful for establishing a zero surface charge density condition on the virtual tube periphery.

Terminal

The **Terminal** node provides a boundary or domain condition for connection to external circuits, to transmission lines, or with a specified voltage or charge.

For more information about the Terminal condition, see Terminal in the documentation for the Electrostatics interface.

For 3D components, enter the **Edge radius** (SI unit: m) for the virtual tube surrounding the edge. The default value is 0.1 m. It is generally assumed that the edge radius is small in comparison to other geometrical measures of the model, such as the edge length and the distances between different edges in the geometry.

To correctly calculate the terminal charge and the corresponding capacitances, if voltage Terminal is used on boundaries between The Electrostatics Interface and The Electrostatics, Boundary Elements Interface, let the dependent variable for The Electrostatics, Boundary Elements Interface be the same as the dependent variable for The Electrostatics Interface and disable Electric Scalar-Scalar Potential Coupling. How to set the name for the dependent variable is described in the Dependent Variables section. Since the Electric Scalar-Scalar Potential Coupling is disabled, add manually a Fully Coupled node to the Stationary Solver below the Solution node in the Solver Configurations.

Zero Charge

The **Zero Charge** node adds the condition that there is zero charge on the boundary so that $\mathbf{n} \cdot \mathbf{D} = 0$. This is the default boundary condition on exterior boundaries and on edges embedded within a Charge Conservation feature.

Electric Scalar-Scalar Potential Coupling

The **Electric Scalar-Scalar Potential Coupling** multiphysics node assures continuity of the electric potential across boundaries between The Electrostatics Interface and The Electrostatics, Boundary Elements Interface. The **Electric Scalar-Scalar Potential Coupling** node is available from the **Multiphysics** menu in the **Physics** toolbar or by right-clicking the **Multiphysics Couplings** node in **Model Builder**, if both The

Electrostatics Interface and The Electrostatics, Boundary Elements Interface are available.

	The Electrostatics Interface and The Electrostatics, Boundary Elements Interface can also be coupled by using the same name for the dependent variable for both interfaces. Then **Electric Scalar-Scalar Potential Coupling** is not needed. How to set the name for the dependent variable is described in the Dependent Variables section.

	When using The Electrostatics, Boundary Elements Interface with iterative solvers, **Right Preconditioning** is preferred. Thus, it is advised to check that **Right Preconditioning** is used when combining the **Electric Scalar-Scalar Potential Coupling** with The Electrostatics, Boundary Elements Interface and other physics interfaces. How to define **Right Preconditioning** is described in Iterative in the *COMSOL Multiphysics Reference Manual*.

BOUNDARY SELECTION

Select **Manual** or **All boundaries** from the **Selection** list. Make additional edits to the list of boundary entity numbers using the Selection toolbar buttons. When **All boundaries** is selected from the **Selection** list, the boundaries exterior to the Electrostatics interface that intersect the exterior boundaries to the Electrostatics, Boundary Elements interface are available in the boundary entity number list.

COUPLED INTERFACES

- Select **Electrostatics, Boundary Elements** as **Primary interface** and **Electrostatics** as **Secondary interface**.

The Electric Currents Interface

The **Electric Currents (ec)** interface ( ), under the **AC/DC>Electric Fields and Currents** branch when adding a physics interface, is used to compute electric field, current, and potential distributions in conducting media under conditions where inductive effects are negligible; that is, when the skin depth is much larger than the studied device.

Depending on the licensed products, stationary, frequency-domain, small-signal analysis, and time-domain modeling are supported in all space dimensions. In the time and frequency domains, capacitive effects are also accounted for.

The physics interface solves a current conservation equation based on Ohm's law using the scalar electric potential as the dependent variable. The interface is typically used to model good conductors for which Ohm's law applies. It is not suitable for modeling poor conductors like air and insulators.

Current Conservation is the main node, which adds the equation for the electric potential and provides a Settings window for defining the electrical conductivity as well as the constitutive relation for the electric displacement field and its associated material properties, such as the relative permittivity.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Current Conservation**, **Electric Insulation** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and current sources. You can also right-click **Electric Currents** to select physics features from the context menu.

Physics-Controlled Mesh

The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the **Sequence type** is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the check box in the **Use** column on the same table row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

Information from the physics, such as the presence of an infinite elements domain or periodic condition, will be used to automatically set up an appropriate meshing sequence.

> 🔷 In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `ec`.

CROSS-SECTION AREA (1D)

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

THICKNESS (2D)

Enter a default value for the **Out-of-plane thickness** d (SI unit: m) (see Equation 3-9). The default value of 1 m is typically not representative for a thin dielectric medium, for example. Instead it describes a unit thickness that makes the 2D equation identical to the equation used for 3D components. See also Change Thickness (Out-of-Plane) (described for the Electrostatics interface).

MANUAL TERMINAL SWEEP SETTINGS

Enter a **Reference impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω.

Select the **Use manual terminal sweep** check box to switch on the sweep and invoke a parametric sweep over the terminals. Enter a **Sweep parameter name** to assign a specific name to the variable that controls the terminal number solved for during the sweep. The default is `PortName`. The generated lumped parameters are in the form of

capacitance matrix elements. The terminal settings must consistently be of either fixed voltage or fixed charge type.

The lumped parameters are subject to **Touchstone file export**. Enter a file path or **Browse** for a file. Select a **Parameter format (value pairs)** for the Touchstone export — **Magnitude and angle (MA)** (the default), **Magnitude in dB and angle (DB)**, or **Real and imaginary parts (RI)**. Select an option from the **If file exists** list — **Overwrite** or **Create new**. Create new is useful when the model is solved multiple times with different settings. Select a **Parameter to export** — **Z** (the default), **Y**, or **S**.

	For most applications it is simpler to sweep over terminals using the Stationary Source Sweep study step. See Stationary Source Sweep in the *COMSOL Multiphysics Reference Manual* and (for more details) Stationary Source Sweep in the *AC/DC Module User's Guide*.

DEPENDENT VARIABLES

The dependent variable is the **Electric potential** V. You can change its name, which changes both the field name and the variable name. If the new name coincides with the name of another electric potential field in the model, the physics interfaces share degrees of freedom. The new name must not coincide with the name of a field of another type or with a component name belonging to some other field.

DISCRETIZATION

Select the shape order for the **Electric potential** dependent variable — **Linear**, **Quadratic** (the default), **Cubic**, **Quartic**, or **Quintic**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

	• Domain, Boundary, Edge, Point, and Pair Nodes for the Electric Currents Interface • Theory for the Electric Currents Interface

	Pacemaker Electrode: Application Library path **COMSOL_Multiphysics/ Electromagnetics/pacemaker_electrode**

Domain, Boundary, Edge, Point, and Pair Nodes for the Electric Currents Interface

The Electric Currents interface has these domain, boundary, edge, point, and pair nodes available from the **Physics** ribbon toolbar (Windows users) or **Physics** context menu (Mac or Linux users). You can also right-click to access the context menu (all users).

> In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

ABOUT THE BOUNDARY CONDITIONS

The exterior and interior boundary conditions listed in Table 3-1 are available. The relevant physics interface condition at interfaces between different media and interior boundaries is continuity; that is,

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

which is the natural boundary condition.

AVAILABLE NODES

These nodes are available for this physics interface, listed in alphabetical order. Also see Table 3-1 for a list of interior and exterior boundary conditions, including edge, point, and pair availability.

- Boundary Current Source
- Contact Impedance
- Current Conservation
- Current Source
- Distributed Impedance
- Electric Insulation
- Electric Point Dipole
- Electric Point Dipole (on Axis)
- Electric Shielding
- External Current Density

- Floating Potential
- Initial Values
- Line Current Source
- Line Current Source (on Axis)
- Normal Current Density
- Piezoresistive Material[1]
- Point Current Source
- Sector Symmetry
- Symmetry Plane (for Electric Field)
- Terminal

[1]This feature is available with the Piezoresistivity, Domain Currents interface, which requires the MEMS Module.

These nodes are described for the Electrostatics interface:

- Change Cross Section
- Change Thickness (Out-of-Plane)
- Electric Potential

- Force Calculation
- Ground
- Periodic Condition

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

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

Table 3-1 lists the interior and exterior boundary conditions available with this physics interface. It also includes edge, point, and pair availability.

TABLE 3-2: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS (INCLUDING EDGE, POINT, AND PAIR AVAILABILITY) FOR THE ELECTRIC CURRENTS INTERFACE

NODE	INTERIOR	EXTERIOR	ALSO AVAILABLE FOR
Boundary Current Source	x		pairs
Contact Impedance	x		pairs
Distributed Impedance	x	x	not applicable
Electric Insulation	x	x	pairs
Electric Potential	x	x	edges, points, and pairs
Electric Shielding	x	x	pairs
Floating Potential	x	x	pairs
Ground	x	x	edges, points, and pairs
Normal Current Density		x	not applicable

TABLE 3-2: INTERIOR AND EXTERIOR BOUNDARY CONDITIONS (INCLUDING EDGE, POINT, AND PAIR AVAILABILITY) FOR THE ELECTRIC CURRENTS INTERFACE

NODE	INTERIOR	EXTERIOR	ALSO AVAILABLE FOR
Periodic Condition		x	not applicable
Symmetry Plane		x	not applicable
Terminal	x	x	pairs, domains

Current Conservation

The **Current Conservation** node adds the continuity equation for the electrical potential and provides an interface for defining the electric conductivity as well as the constitutive relation and the relative permittivity for the displacement current.

MATERIAL TYPE

The **Material type** setting decides how materials behave and how material properties are interpreted when the mesh is deformed. Select **Solid** for materials whose properties change as functions of material strain, material orientation, and other variables evaluated in a material reference configuration (material frame). Select **Non-solid** for materials whose properties are defined only as functions of the current local state at each point in the spatial frame, and for which no unique material reference configuration can be defined. Select **From material** to pick up the corresponding setting from the domain material on each domain.

CONSTITUTIVE RELATION JC-E

By default, the **Electrical conductivity** σ (SI unit: S/m) for the media is defined **From material**. Or select **User defined** or **Linearized resistivity**.

User Defined

For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** depending on the characteristics of the electrical conductivity, and then enter values or expressions for the electrical conductivity σ in the field or matrix. The default is 0 S/m. If type of temperature dependence is used other than a linear temperature relation, enter any expression for the conductivity as a function of temperature.

Linearized Resistivity

Select **Linearized resistivity** for a temperature-dependent conductivity (this occurs in, for example, Joule heating, and is also called resistive heating). The equation describing the conductivity:

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

where ρ_0 is the resistivity at the reference temperature T_{ref}, and α is the temperature coefficient of resistance, which describes how the resistivity varies with temperature.

The default **Reference resistivity** ρ_0 (SI unit: $\Omega \cdot$m), **Reference temperature** T_{ref} (SI unit: K), and **Resistivity temperature coefficient** α (SI unit: $1/$K) are taken **From material**, which means that the values are taken from the domain (or boundary) material. T is the current temperature, which can be a value that is specified as a model input or the temperature from a heat transfer interface. The definition of the temperature field is in the **Model Inputs** section.

To specify other values for any of these properties, select **User defined** from the list and then enter a value or expression for each. The default values are:

- 1 $\Omega \cdot$m for the Reference resistivity
- 273.15 K for the Reference temperature, and
- 0 1/K for the Resistivity temperature coefficient

CONSTITUTIVE RELATION D-E

Select a **Dielectric model** to describe the macroscopic properties of the medium (relating the electric displacement **D** with the electric field **E**) and the applicable material properties, such as the relative permittivity. For a description of the constitutive relations **Relative permittivity**, **Polarization**, and **Remanent electric displacement**, see **Constitutive Relation D-E** as described for the Charge Conservation node for the Electrostatics interface. The constitutive relations specific to Electric Currents are:

- **Dielectric losses**: uses the constitutive relation $\mathbf{D} = \varepsilon_0(\varepsilon' - j\varepsilon'')\mathbf{E}$. Specify that the **Relative permittivity (real part)** ε' (dimensionless) and the **Relative permittivity (imaginary part)** ε'' (dimensionless) must be taken **From material** or be **User defined**. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix. The default is 1.

- **Loss tangent, loss angle**: uses the constitutive relation $\mathbf{D} = \varepsilon_0 \varepsilon'(1 - j\tan\delta)\mathbf{E}$. Specify the **Relative permittivity (real part)** ε' (dimensionless) and **Loss angle** δ (SI unit: rad).

- **Loss tangent, dissipation factor**: uses the constitutive relation $\mathbf{D} = \varepsilon_0 \varepsilon'(1 - j\tan\delta)\mathbf{E}$. Specify the **Relative permittivity (real part)** ε' (dimensionless) and the **Dissipation factor** $\tan\delta$ (dimensionless).

Initial Values

The **Initial Values** node adds an initial value for the electric potential that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver. If more than one set of initial values is required, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and current sources. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

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

External Current Density

The **External Current Density** node adds an externally generated current density $\mathbf{J}_e$, which appears in Ohm's law

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

and in the equation that the physics interface defines.

The external current density does not contribute to the losses (due to Joule heating), since there is no electric field associated with it. To include the contribution to the losses from the external current density, select the **Add contribution of the external current density to the losses** check box. Then select an option from the **External losses** list — **From domain conductivity** (the default) or **User defined**. If **From domain conductivity** is selected, the heat source is computed using the conductivity specified in the material model feature (such as **Current Conservation**) that is applied in the domain. For **User defined**, enter a value for Q_e (SI unit: W/m^3) to specify a user-defined heat source.

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** on the **Physics** toolbar. For more information see

Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

EXTERNAL CURRENT DENSITY

Based on space dimension, enter the coordinates (**x**, **y**, and **z** for 3D components, for example) of the **External current density** $\mathbf{J}_e$ (SI unit: A/m^2). The defaults are 0 A/m^2.

Current Source

The **Current Source** node adds a distributed current source Q_j in the equation that the physics interface defines. Use this node with caution as it can violate the current conservation law that is inherent in Maxwell-Ampère's law.

Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** on the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

CURRENT SOURCE

Enter a value or expression for the **Current source** Q_j (SI unit: A/m^3). The default is 0 A/m^3.

Electric Insulation

The **Electric Insulation** node, which is the default boundary condition, adds electric insulation as the boundary condition:

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

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

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

Floating Potential

The **Floating Potential** node is used when modeling a metallic electrode at floating potential. It applies a constant voltage V_0 on the boundary (for domain features, this is the boundary enclosing the selected domain), such that the total normal electric current density $\mathbf{J}$ equals a specific current I_0:

$$V = V_0$$

$$\int_{\partial\Omega} (-\mathbf{n} \cdot \mathbf{J})dS = I_0,$$

where Ω represents the boundary and $\mathbf{n}$ refers to the surface normal. The constant boundary voltage implies the tangential electric field equals zero; the electric field will be perpendicular to the boundary:

$$-\nabla_t V = \mathbf{E}_t = 0$$
$$\mathbf{n} \times \mathbf{E} = 0.$$

In case of $I_0 = 0$ (the default case), the boundary will behave as an unconnected perfect conductor (a floating equipotential). This is a good approximation when the conductivity of the electrode is many orders of magnitude larger than that of the surrounding medium. Although locally the current density may vary, the total current entering or leaving the boundary equals zero.

If the floating potential touches a point, boundary or domain feature that is not floating (a **Terminal** or **Ground** feature), the floating potential will acquire that feature's potential. If the floating potential is set to a certain current, or connected to a circuit, it behaves like a terminal.

The **Harmonic Perturbation** subnode (it is of the exclusive type) is available from the context menu (right-click the parent node) or on the **Physics** toolbar, click the **Attributes** menu and select **Harmonic Perturbation**. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

FLOATING POTENTIAL

The Floating potential identifier text area shows the unique identifier for the floating potential feature. It is used to identify the global variables created by the node, such as the voltage. The **Floating potential group** check box on the **Settings** window for **Floating Potential** controls how potentials are assigned to boundary segments in the feature's boundary selection. If this check box is not selected, a single potential is applied to all

boundaries in the selection. If the check box is selected, each group of contiguous boundaries in the selection is given a unique potential. This simplifies the setup of models with many floating electrodes. The values of the potential at each group of boundaries are then made available in postprocessing, collected in a vector variable.

The following options are not available if the **Floating potential group** check box is selected. Select a specification for the **Floating potential current** — choose **User defined** (the default) to specify a total **Current** I_0 (SI unit: A) that flows from the electrode. The default is 0 A, corresponding to an unconnected electrode. Select **Circuit** to connect the floating potential to an Electrical Circuit interface.

Initial values
The initial values subsection is used to provide initial values for the internal state variables in the Floating Potential. These settings can be useful to provide an initial guess to the nonlinear solver or the initial value in a time-dependent study when the initial values of the other dependent variables (such as the electric potential) are computed from a separate physics or study. In most cases, the default values are appropriate.

Enter an **Initial value for voltage** V_{init} (SI unit: V). The default is 0 V.

CONSTRAINT SETTINGS
To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

ℚ	Settings and Properties Windows for Feature Nodes in the *COMSOL Multiphysics Reference Manual*

Boundary Current Source

The **Boundary Current Source** node adds a current source Q_j on the boundary.

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = Q_j$$

It is applicable to interior boundaries that represent either a source or a sink of current. Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or click **Harmonic Perturbation** on the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

BOUNDARY CURRENT SOURCE

Enter a value or expression for the **Boundary current source** Q_j (SI unit: A/m^2). The default is 0 A/m^2.

Normal Current Density

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

$$-\mathbf{n} \cdot \mathbf{J} = J_n$$

Or alternatively, as a current density $\mathbf{J}_0$:

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

The normal current density is positive when the current flows inward in the domain. Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** on the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

NORMAL CURRENT DENSITY

Select a **Type** — **Inward current density** (the default) or **Current density**.

- For **Inward current density** enter a value or expression for the **Normal current density** J_n (SI unit: A/m^2). Use a positive value for an inward current flow or a negative value for an outward current flow. The default is 0 A/m^2.

- For **Current density** enter values or expressions for the components of the **Current density** $\mathbf{J}_0$ (SI unit: A/m^2). The defaults are 0 A/m^2.

Distributed Impedance

The **Distributed Impedance** node adds a distributed impedance boundary condition to a model.

The **Harmonic Perturbation** subnode (it is of the exclusive type) is available from the context menu (right-click the parent node) or on the **Physics** toolbar, click the **Attributes** menu and select **Harmonic Perturbation**. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

Use this boundary condition to model a thin sheet of a resistive material connected to a reference potential V_{ref}.

The layer impedance can be specified either with the bulk material conductivity σ_s, the relative permittivity ε_r and layer thickness d_s, or directly with the surface resistance ρ_s and capacitance C_s. Assuming DC currents, the equation is:

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

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

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:

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

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

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

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

DISTRIBUTED IMPEDANCE

Enter the reference potential V_{ref} (SI unit: V). The default is 0 V.

Select a potentially complex-valued **Layer specification** — **Thin layer** (the default) or **Surface impedance**.

- For **Thin layer**, enter values or expressions for the:
  - **Surface thickness** d_s (SI unit: m). The default is $5 \cdot 10^{-3}$ m (5 mm).
  - **Electrical conductivity** σ (SI unit: S/m) and **Relative permittivity** ε_r (dimensionless). The defaults take values **From material**. For **User defined**, enter different values or expressions. The default electrical conductivity is $1 \cdot 10^{-2}$ S/m and the default relative permittivity is 1.

- For **Surface impedance**, enter values or expressions for the **Surface resistance** ρ_s (SI unit: $\Omega \cdot \text{m}^2$) and the **Surface capacitance** C_s (SI unit: F/m^2). The default surface impedance is $1 \cdot 10^{-8}$ $\Omega \cdot \text{m}^2$ and the default surface capacitance is 0 F/m^2.

Terminal

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

> ⚠️ In most cases, it is not recommended to include the interior of good conductors in the simulation. When using the **Terminal** on the domain level, the interior unknowns for the electric potential are replaced by a single constant, thus avoiding numerical problems arising from a very high material contrast. If one instead uses the **Terminal** on the boundary level, it is recommended to exclude the interior of the associated conductor from the domain selection for the main Electric Currents node.

The **Terminal** node also enables the computation of the lumped parameter of the system, such as the impedance.

The **Harmonic Perturbation** subnode (it is of the exclusive type) is available from the context menu (right-click the parent node) or on the **Physics** toolbar, click the **Attributes** menu and select **Harmonic Perturbation**. The **Harmonic Perturbation** subnode only has effect if **Current** or **Voltage** are selected as terminal type. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual.*

An operation named as **Split by Connectivity** (🖊) is available by right-clicking the parent node. The operation is useful when there are several terminal domains or boundaries that are not geometrically connected. Clicking the operation node will automatically generate several **Terminal** nodes where each selection is geometrically connected.

TERMINAL

Specify the terminal's properties. The **Terminal name** identifies the terminal and the variables it creates. The name should be numeric for sweeps to work properly.

Select a **Terminal type** — **Current** (the default), **Voltage**, **Circuit**, **Terminated,** or **Power**.

> ❗ The **Circuit** and **Power** types cannot be used together with a terminal sweep.

Select:

- **Current** to specify a total current flowing from the terminal. Enter the current I_0 (SI unit: A). The default is 0 A for an open circuit.

- **Voltage** to specify a fixed electric potential at the terminal. Enter the value of the electric potential V_0 (SI unit: V). The default is 1 V.

- **Circuit** to specify a terminal connected to an Electrical Circuit interface that models an external circuit.

- **Terminated** to connect the terminal to an impedance that might represent a load or a transmission line. When **Terminated** is selected, the scattering parameters (S-parameters) are computed.

  - Enter a **Terminal power** P_0 (SI unit: W) to specify the input power at this terminal from the transmission line. This excitation can be toggled using a terminal sweep. The default is 1 W.

  - Select an option from the **Characteristic impedance from** list to define the value of the impedance — **Physics interface** (the default) or **User defined**. For **Physics interface** the **Reference impedance** Z_{ref} defined on the **Settings** window for the physics interface under Sweep Settings is used. For **User defined** enter a specific impedance Z_{ref} (SI unit: Ω) for this terminal. The default is 50 Ω.

- **Power** to specify a fixed electrical power for the terminal. The power is computed as the product of the terminal current and voltage and, in frequency domain, is the cycle-averaged power. When **Power** is selected:

  - Enter a **Terminal power** P_0 (SI unit: W) to specify the input power at this terminal. The default is 1 W.

❗	Using Power excitation causes the equation system to become nonlinear. COMSOL Multiphysics automatically adjusts the solver settings when Power is selected.
	Be aware that, in general, the values of the voltage and the current might not uniquely be determined by this constraint and may depend on the initial value provided for the current and voltage variables. Particularly in the frequency domain, the absolute phase of the quantities can be arbitrary. This indeterminacy can have an impact on the solution process. See Compile Equations in the *COMSOL Multiphysics Reference Manual* for a possible solution to this problem.

Initial Values

The initial values subsection is used to provide initial values for the internal state variables in the Terminal. These settings can be useful to provide an initial guess to the nonlinear solver or the initial value in a time-dependent study when the initial values of the other dependent variables (such as the electric potential) are computed from a separate physics or study. In most cases, the default values are appropriate.

The **Initial value for voltage** is available for all Terminal types except Voltage. Enter an **Initial value for voltage** V_{init} (SI unit: V). The default is 0 V.

The **Initial value for current** is available for Power only. In this case, it provides an initial guess for the nonlinear solver, and should not be set to zero. The initial value provided may affect the solution. Enter an **Initial value for current** I_{init} (SI unit: A). The default is 1 mA.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

Electric Shielding

The **Electric Shielding** node provides an electric shielding boundary condition. Use this node to model a thin layer of a highly conductive medium that shields the electric field. The sheet has the electrical conductivity σ_s and the surface thickness d.

The condition is represented by the following equation for interior boundaries and (setting $\mathbf{J}_2 = 0$) exterior boundaries assuming DC currents

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = -\nabla_t \cdot d(\sigma_s \nabla_t V)$$

For the frequency domain and Time Dependent study types, also displacement currents are accounted for via the bulk relative permittivity of the sheet; ε_{rs} and the conservation laws change to:

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

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

Use this boundary condition when approximating a thin domain with a boundary to reduce the number of mesh elements. The **Electric Field** and **Conduction Current** sections are the same as for Current Conservation.

THICKNESS

Enter a value or expression for the **Surface thickness** d_s (SI unit: m). The default is 1 cm.

Contact Impedance

Use the **Contact Impedance** node on interior boundaries to model a thin layer of resistive material. It can also be added as a pair using a **Pair Contact Impedance** node. The feature allows specifying the contact impedance either by entering the properties of the material together with the layer thickness, or by entering the impedance properties of the thin layer directly.

The feature, in stationary studies, applies the following conditions that relate the normal electric current density with the jump in the electric potential:

$$\mathbf{n} \cdot \mathbf{J}_1 = \frac{\sigma}{d_s}(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \frac{\sigma}{d_s}(V_2 - V_1)$$

$$\mathbf{n} \cdot \mathbf{J}_1 = \frac{1}{\rho_s}(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \frac{1}{\rho_s}(V_2 - V_1)$$

The first two equations refer to a layer impedance specified using the bulk material conductivity σ_s and the layer thickness d_s, while the last two equations refer to the case in which the surface resistance ρ_s is specified. The indices 1 and 2 refer to the two sides of the boundary. These parameters work the same as with Distributed Impedance.

For the frequency domain and Time Dependent study types, this boundary condition is slightly more sophisticated and accounts also for capacitive coupling. The bulk material relative permittivity ε_r or the layer capacitance C_s can be specified.

The equations used in Frequency Domain studies are:

$$\mathbf{n} \cdot \mathbf{J}_1 = \frac{(\sigma + j\omega\varepsilon_0\varepsilon_r)}{d_s}(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \frac{(\sigma + j\omega\varepsilon_0\varepsilon_r)}{d_s}(V_2 - V_1)$$

$$\mathbf{n} \cdot \mathbf{J}_1 = \left(\frac{1}{\rho_s} + j\omega C_s\right)(V_1 - V_2)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \left(\frac{1}{\rho_s} + j\omega C_s\right)(V_2 - V_1)$$

In Time Dependent studies:

$$\mathbf{n} \cdot \mathbf{J}_1 = \frac{1}{d_s}\left(\sigma(V_1 - V_2) + \varepsilon_0\varepsilon_r\frac{\partial}{\partial t}(V_1 - V_2)\right)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \frac{1}{d_s}\left(\sigma(V_2 - V_1) + \varepsilon_0\varepsilon_r\frac{\partial}{\partial t}(V_2 - V_1)\right)$$

$$\mathbf{n} \cdot \mathbf{J}_1 = \left(\frac{1}{\rho_s}(V_1 - V_2) + C_s\frac{\partial}{\partial t}(V_1 - V_2)\right)$$

$$\mathbf{n} \cdot \mathbf{J}_2 = \left(\frac{1}{\rho_s}(V_2 - V_1) + C_s\frac{\partial}{\partial t}(V_2 - V_1)\right)$$

CONTACT IMPEDANCE

Select a potentially complex-valued **Layer specification** — **Thin layer** (the default) or **Surface impedance**.

- For **Thin layer**, enter values or expressions for the:
  - **Surface thickness** d_s (SI unit: m). The default is $5 \cdot 10^{-3}$ m (5 mm).
  - **Electrical conductivity** σ (SI unit: S/m) and **Relative permittivity** ε_r (dimensionless). The defaults take values **From material**. For **User defined**, enter different values or expressions. The default electrical conductivity is $1 \cdot 10^{-2}$ S/m and the default relative permittivity is 1.

- For **Surface impedance**, enter values or expressions for the **Surface resistance** ρ_s (SI unit: $\Omega \cdot m^2$) and the **Surface capacitance** C_s (SI unit: F/m^2). The default surface impedance is $1 \cdot 10^{-8}$ $\Omega \cdot m^2$ and the default surface capacitance is 0 F/m^2.

▦　*Thin-Film Resistance*: Application Library path **COMSOL_Multiphysics/ Electromagnetics/thin_film_resistance**

Sector Symmetry

Select **Sector Symmetry** at interfaces between rotating objects where sector symmetry is used. It is only available for pairs. A default subnode is added. Right-click to select additional features from the **Fallback Features** submenu. In 2D, this feature assumes rotation around the origin.

> This feature is always used in conjunction with a Periodic Condition on adjacent radial sector boundaries. Note that the same periodic condition feature cannot be used on both sides of where the sector symmetry boundaries connect with the periodic boundaries. At least two periodic condition features are required for the model to compute correctly.

PAIR SELECTION

When using nonconforming meshes on the source and destination of a pair, for numerical stability, a finer mesh should be applied on the destination side for any pair with a condition that imposes a coupling or a constraint across the pair. The sector symmetry feature falls into this category.

SECTOR SETTINGS

Enter the **Number of sectors (<50)** n_{sect}. The default is 2.

Select a **Type of periodicity** — **Continuity** (the default) or **Antiperiodicity**.

Based on space dimension, enter values or expressions in the table for the **Axis of rotation** $\mathbf{a}_{\mathrm{rot}}$.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

Line Current Source

The **Line Current Source** node adds a line source to edges in 3D components and to points in 2D and 2D axisymmetric components. The line source represents electric current per unit length.

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

LINE CURRENT SOURCE

Enter a value or expression to apply a **Line current source** Q_j (SI unit: A/m). This source represents electric current per unit length. The default is 0 A/m.

🔍 Line Current Source (on Axis) for 2D axisymmetric components.

Line Current Source (on Axis)

The **Line Current Source (on Axis)** node adds a line source to boundaries in 2D axisymmetric components. The line source represents electric current per unit length.

LINE CURRENT SOURCE (ON AXIS)

Enter a value or expression to apply a **Line current source** Q_j (SI unit: A/m) to boundaries. This source represents electric current per unit length.

🔍 Line Current Source

Point Current Source

The **Point Current Source** node adds a point source and represents an electric current flowing out of the point. Add point sources to 3D components from the **Points** menu. Add a contribution as a **Harmonic Perturbation** by right-clicking the parent node or clicking **Harmonic Perturbation** on the **Physics** toolbar. For more information see Harmonic Perturbation — Exclusive and Contributing Nodes in the *COMSOL Multiphysics Reference Manual*.

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

POINT CURRENT SOURCE

Enter a value or expression to apply a **Point current source** Q_j (SI unit: A) to points. This source represents an electric current flowing out of the point.

🔍   • Line Current Source for 2D components
     • Point Current Source (on Axis) for 2D axisymmetric components

Point Current Source (on Axis)

The **Point Current Source (on Axis)** node adds a point source and represents an electric current flowing out of the point in 2D axisymmetric components.

POINT SELECTION

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

POINT CURRENT SOURCE

Enter a value or expression to apply a **Point current source** Q_j (SI unit: A) to points. This source represents an electric current flowing out of the point.

🔍   • Point Current Source for 3D components
     • Line Current Source for 2D components

Electric Point Dipole

The **Electric Point Dipole** node, available for 2D and 3D components, represents the limiting case of zero separation distance between two equally strong point current sources and current sinks of opposing signs while maintaining the product between

separation distance and source strength at a fixed value (*P)*. The positive direction is from the current sink to the current source.

DIPOLE SPECIFICATION

Select a **Dipole specification** — **Magnitude and direction** (the default) or **Dipole moment**.

DIPOLE PARAMETERS

Under **Dipole Specification**:

- For **Magnitude and direction** enter coordinates for the **Electric current dipole moment direction** $\mathbf{n}_p$ and the **Electric current dipole moment, magnitude** p (SI unit: A·m).

- For **Dipole moment** enter the components of the **Electric current dipole moment p** (SI unit: A·m).

Q | Electric Point Dipole (on Axis)

Electric Point Dipole (on Axis)

The **Electric Point Dipole (on Axis)**, available for 2D axisymmetric components, represents the limiting case of zero separation distance between two equally strong point current sources and current sinks of opposing signs while maintaining the product between separation distance and source strength at a fixed value (*P*). The positive direction is from the current sink to the current source.

ELECTRIC POINT DIPOLE (ON AXIS)

Enter the **Electric current dipole moment in z direction** p_z (SI unit: A·m).

Q | Electric Point Dipole

Piezoresistive Material

The **Piezoresistive Material** is normally used together with a **Piezoresistive Effect, Domain Currents** multiphysics coupling node. The node is added by default to the **Electric Currents** interface when adding a **Piezoresistivity, Domain Currents** predefined

multiphysics coupling interface. It is also available from the context menu (right-click the **Electric Currents** interface parent node) or from the **Physics** toolbar.

> ⚠ When the **Piezoresistive Material** node is added to the Electric Currents interface in the absence of an active **Piezoelectric Effect, Domain Currents** multiphysics coupling node, the material behaves similarly to a Current Conservation node, with electric properties corresponding to the relative permittivity and electrical conductivity entered. The piezoresistive effect is *not* included in the corresponding equation system.

PIEZORESISTIVE MATERIAL PROPERTY

This node should be used together with an active **Piezoresistive Effect, Domain Currents** multiphysics coupling node. Select a **Constitutive model** — **Piezoresistance form** or **Elastoresistance form**. For each of the following, the default uses values **From material**. For **User defined** enter other values in the matrix or field.

- Specify a **Electrical conductivity, zero stress** (SI unit: S/m). This typically comes from the material added under the Materials node.

- For **Piezoresistance form**, select a **Piezoresistance coupling matrix** Π_l (SI unit: $m^4/(s \cdot A^2)$; note that this is equivalent to $\Omega \cdot m / Pa$).

- For a **Elastoresistance form**, select an **Elastoresistance coupling matrix** M_l (SI unit: $\Omega \cdot m$).

The Electrical Circuit Interface

The **Electrical Circuit (cir)** interface (![icon]), found under the **AC/DC** branch (![icon]) when adding a physics interface, is used to model currents and voltages in circuits including voltage and current sources, resistors, capacitors, inductors, and semiconductor devices. Models created with the Electrical Circuit interface can include connections to distributed field models. The physics interface supports stationary, frequency-domain and time-domain modeling and solves Kirchhoff's conservation laws for the voltages, currents and charges associated with the circuit elements.

When this physics interface is added, it adds a default **Ground Node** feature and associates that with node zero in the electrical circuit.

> **!** Circuit nodes are nodes in the electrical circuit (electrical nodes) and should not be confused with nodes in the Model Builder tree of the COMSOL Multiphysics software. Circuit node names are not restricted to numerical values but can contain alphanumeric characters.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `cir`.

RESISTANCE IN PARALLEL TO PN JUNCTIONS

For numerical stability, a large resistance is added automatically in parallel to the pn junctions in diodes and BJT devices. Enter a default value for the **Resistance in parallel to pn junctions** R_j (SI unit: Ω). The default value is $1 \cdot 10^{12}$ Ω.

CREATE UNIQUE NODES FOR NEW DEVICES

When this setting is selected (the default), newly added devices will be assigned unused node names. The devices will be disconnected from the rest of the circuit and the nodes should be updated to reflect the actual circuit connections. When this setting is

deselected, new devices will be connected to the lowest-numbered nodes starting from 0.

	• Theory for the Electrical Circuit Interface
Q	• Connecting to Electrical Circuits

ELECTRICAL CIRCUIT TOOLBAR

The following nodes are available from the **Electrical Circuit** ribbon toolbar (Windows users), **Electrical Circuit** context menu (Mac or Linux users), or right-click to access the context menu (all users):

	For step-by-step instructions and general documentation
💡	descriptions, this is the **Electrical Circuit** toolbar.

- Ground Node
- Voltmeter
- Ampère Meter
- Resistor
- Capacitor
- Inductor
- Voltage Source
- Current Source
- Diode
- Switch
- Voltage-Controlled Voltage Source[1]
- Voltage-Controlled Current Source[1]
- Current-Controlled Voltage Source[1]
- Current-Controlled Current Source[1]

- Subcircuit Definition
- Subcircuit Instance
- Mutual Inductance
- Transformer
- NPN BJT and PNP BJT[2]
- n-Channel MOSFET and p-Channel MOSFET[2]
- External I vs. U[3]
- External U vs. I[3]
- External I-Terminal[3]
- SPICE Circuit Import
- SPICE Circuit Export

[1] Selected from the **Dependent Sources** submenu when you right-click main node.

[2] Selected from the **Transistors** submenu when you right-click main node.

[3] Selected from the **External Couplings** submenu when you right-click main node.

Ground Node

The **Ground Node** (⏚) feature adds a ground node with the default node number zero to the electrical circuit. This is the default node in the Electrical Circuit interface. More ground nodes can be added but those must have unique node numbers and are by default given higher node numbers.

GROUND CONNECTION

Set the **Node name** for the ground node in the circuit. The convention is to use 0 (zero) for the ground node. If adding more ground nodes, each must have a unique node name (number).

Voltmeter

The **Voltmeter** (ⓥ) feature connects a voltmeter (voltage measurement device) between two nodes in the electrical circuit. A voltmeter behaves electrically as an open circuit. The voltmeter node adds a Probe sampling the voltage across it.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the resistor.

Ampère Meter

The **Ammeter** (ⓐ) feature connects an ammeter (current measurement device) between two nodes in the electrical circuit. An ammeter behaves electrically as a short circuit. The ammeter node adds a Probe sampling the current through it.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the resistor.

DEVICE PARAMETERS

Enter the **Resistance** of the resistor.

Resistor

The **Resistor** (▭) feature connects a resistor between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the resistor.

DEVICE PARAMETERS

Enter the **Resistance** of the resistor.

Capacitor

The **Capacitor** (⊣⊢) feature connects a capacitor between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the capacitor.

DEVICE PARAMETERS

Enter the **Capacitance** of the capacitor.

Inductor

The **Inductor** (₀₀₀) feature connects an inductor between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the inductor.

DEVICE PARAMETERS

Enter the **Inductance** of the inductor.

Voltage Source

The **Voltage Source** (⊕) feature connects a voltage source between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the voltage source. The first node represents the positive reference terminal.

Enter the **Source type** that should be adapted to the selected study type. It can be **General source**, **AC-source**, or a time-dependent **Sine source** or **Pulse source**. Depending on the choice of source, also specify the following parameters:

- For a General source, the **Voltage V_{src}** (default value: 1 V). General sources are active in Stationary, Time-Dependent and Frequency Domain studies.

- For an AC-source: the **Voltage V_{src}** (default value: 1 V) and the **Phase** Θ (default value: 0 rad). AC-sources are active in Frequency Domain studies only.

- For a sine source: the **Voltage V_{src}** (default value: 1 V), the **Offset V_{off}** (default value: 0 V), the **Frequency** (default value: 1 kHz), and the **Phase** Θ (default value: 0 rad). The sine sources are active in Time-Dependent studies and also in Stationary studies, providing that a value for t has been provided as a model parameter or global variable.

- For a pulse source: the **Voltage V_{src}** (default value: 1 V), the **Offset V_{off}** (default value: 0 V), the **Delay t_d** (default value: 0s), the **Rise time t_r** and **Fall time t_f** (default values: 0 s), the **Pulse width p_w** (default value: 1 µs), and the **Period T_{per}** (default value: 2 µs). The pulse sources are active in Time-Dependent studies and also in Stationary studies, providing that a value for t has been provided as a model parameter or global variable.

All values are peak values rather than RMS.

📋	For the AC source, the frequency is a global input set by the solver. AC sources should be used in Frequency-domain studies only. Do not use the **Sine source** unless the model is time dependent.

Current Source

The **Current Source** (ⓦ) feature connects a current source between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the current source. The first node represents the positive reference terminal from where the current flows through the source to the second node.

Enter the **Source type** that should be adapted to the selected study type. It can be **General source**, **AC-source**, or a time-dependent **Sine source** or **Pulse source**. Depending on the choice of source, also specify the following parameters:

- For a General source, the **Current i$_{src}$** (default value: 1 A). General sources are active in Stationary, Time-Dependent and Frequency Domain studies.

- For an AC-source: the **Current i$_{src}$** (default value: 1 A) and the **Phase** Θ (default value: 0 rad). AC-sources are active in Frequency Domain studies only.

- For a sine source: the **Current i$_{src}$** (default value: 1 A), the **Offset i$_{off}$** (default value: 0 A), the **Frequency** (default value: 1 kHz), and the **Phase** Θ (default value: 0 rad). The sine sources are active in Time-Dependent studies and also in Stationary studies, providing that a value for t has been provided as a model parameter or global variable.

- For a pulse source: the **Current i$_{src}$** (default value: 1 A), the **Offset i$_{off}$** (default value: 0 A), the **Delay t$_d$** (default value: 0 s), the **Rise time t$_r$** and **Fall time t$_f$** (default values: 0 s), the **Pulse width p$_w$** (default value: 1 μs), and the **Period T$_{per}$** (default value: 2 μs). The pulse sources are active in Time-Dependent studies and also in Stationary studies, providing that a value for t has been provided as a model parameter or global variable.

All values are peak values rather than RMS.

📋 For the AC source, the frequency is a global input set by the solver. AC sources should be used in frequency-domain studies only. Do not use the **Sine source** unless the model is time dependent.

Voltage-Controlled Voltage Source

The **Voltage-Controlled Voltage Source** (⬦) feature connects a voltage-controlled voltage source between two nodes in the electrical circuit. A second pair of nodes define the input control voltage.

NODE CONNECTIONS

Specify four **Node names**: the first pair for the connection nodes for the voltage source and the second pair defining the input control voltage. The first node in a pair represents the positive reference terminal.

There are two options to define the relationship between the control voltage and resulting voltage. The **Use gain** method defines the resulting voltage to be the control voltage multiplied by the gain. The **Custom expression** method can define the relationship with an arbitrary expression.

Voltage-Controlled Current Source

The **Voltage-Controlled Current Source** (◈) feature connects a voltage-controlled current source between two nodes in the electrical circuit. A second pair of nodes define the input control voltage.

NODE CONNECTIONS

Specify four **Node names**: the first pair for the connection nodes for the current source and the second pair defining the input control voltage. The first node in a pair represents the positive voltage reference terminal or the one from where the current flows through the source to the second node.

DEVICE PARAMETERS

There are two options to define the relationship between the control voltage and resulting current. The **Use gain** method defines the resulting current to be the control voltage multiplied by the gain (SI units: S). The **Custom expression** method can define the relationship with an arbitrary expression.

Current-Controlled Voltage Source

The **Current-Controlled Voltage Source** (◈) feature connects a current-controlled voltage source between two nodes in the electrical circuit. The input control current is the one flowing through a two-pin device.

NODE CONNECTIONS

Set two **Node names** for the connection nodes for the voltage source. The first node in a pair represents the positive reference terminal.

DEVICE PARAMETERS

There are two options to define the relationship between the control current and resulting voltage. The **Use gain** method defines the resulting voltage to be the control current multiplied by the gain (SI units: Ω). The **Custom expression** method can define the relationship with an arbitrary expression.

Current-Controlled Current Source

The **Current-Controlled Current Source** (◇) feature connects a current-controlled current source between two nodes in the electrical circuit. The input control current is the one flowing through a named device that must be a two-pin device.

NODE CONNECTIONS

Specify two **Node names** for the connection nodes for the current source. The first node in a pair represents the positive reference terminal from where the current flows through the source to the second node.

DEVICE PARAMETERS

There are two options to define the relationship between the control current and resulting current. The **Use gain** method defines the resulting current to be the control current multiplied by the gain. The **Custom expression** method can define the relationship with an arbitrary expression.

Switch

The **Switch** (—-) feature is used to connect or disconnect the conducting path in a circuit under specific conditions.

NODE CONNECTIONS

Specify two **Node names** for the connection nodes for the current source. The first node in a pair represents the positive reference terminal from where the current flows through the source to the second node.

SWITCH CONDITIONS

There are three types of conditions, **Voltage controlled**, **Current controlled**, and **Custom expressions**. For each type of condition there are two conditions, one for turn on and one for turn off. The on condition is true if the **On condition** expression is larger than zero, while the off condition is true if the **Off condition** is less than zero.

The **Initial state** list has three options, **Use on condition**, **Use off condition**, and **Boolean expression**. The two former options mean that the switch will have an initial state matching to the on or off condition. The third option makes the switch's initial state match a custom Boolean expression. Separating on, off, and initial states makes the switch more flexible and can support Schmitt-trigger style switches and various latches.

For the **Voltage controlled** switch, it is necessary to specify two nodes that defines the voltage `sens.v` that the switch state depends on. The conditions must be written as a function of this variable. Similarly, for the **Current controlled** switch it is necessary to specify a two-pin device that defines the current `sens.i` that the switch state depends on.

SWITCH PARAMETERS

When the switch is in the on state it has a non-zero resistance specified by the **On resistance** expression. For the off state no current flows through the switch (infinite resistance). There is also a transition time for the switch to turn on and off set by the **Switching time** expression. The switch triggers an implicit event that updates a discrete state variable (with `suffix _state`).

Subcircuit Definition

The **Subcircuit Definition** (🔲) feature is used to define subcircuits, which can be inserted as devices into the main circuit using Subcircuit Instance nodes. Create the subcircuit by adding subnodes to the **Subcircuit Definition** node, either by using the **Physics** toolbar, or by right-clicking the **Subcircuit Definition**.

SUBCIRCUIT PINS

Define the **Pin names** at which the subcircuit connects to the main circuit or to other subcircuits when referenced by a Subcircuit Instance node. The **Pin names** refer to circuit nodes in the subcircuit. The order in which the **Pin names** are defined is the order in which they are referenced by a Subcircuit Instance node. The devices constituting the subcircuit should be connected only to the subcircuit's pins and to themselves.

INPUT PARAMETERS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**. Specify input parameters to a subcircuit that can be changed from a subcircuit instance. These input parameters can be used in all expression-style edit fields that affect the parameters of a device, for example, resistance, capacitance, and current gain. In this way, a subcircuit can represent a parameterized custom device model.

Subcircuit Instance

The **Subcircuit Instance** (🔲) feature represents an instance of a subcircuits defined by a Subcircuit Definition feature.

NODE CONNECTIONS

Select the **Name of subcircuit link** from the list of defined subcircuits in the circuit model and the circuit **Node names** at which the subcircuit instance connects to the main circuit or to another subcircuit if used therein.

INPUT PARAMETERS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**. Specify input parameters to a subcircuit that can be changed from a subcircuit instance. These input parameters can be used in all expression-style edit fields that affect the parameters of a device, for example, resistance, capacitance, and current gain. In this way, a subcircuit can represent a parameterized custom device model.

NPN BJT and PNP BJT

The **NPN BJT** (⊕) and the **PNP BJT** (⊕) device models are large-signal models for bipolar junction transistors (BJT). It is an advanced device model and no thorough description and motivation of the many input parameters are attempted here. Many device manufacturers provide model input parameters for this BJT model. For any particular make of BJT, the device manufacturer should be the primary source of information.

NODE CONNECTIONS

Specify three **Node names** for the connection nodes for the **BJT** device. These represent the *collector*, *base*, and *emitter* nodes for the **NPN** transistor, and the *emitter*, *base*, and *collector* nodes for the **PNP** transistor.

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular BJT, the device manufacturer should be the primary source of information.

> The interested reader is referred to Ref. 1 for more details on semiconductor modeling within circuits.
>
> For an explanation of the **Model Parameters** see Bipolar Transistors.

n-Channel MOSFET and p-Channel MOSFET

The **n-Channel MOSFET** ( ) and the **p-Channel MOSFET** ( ) device models are large-signal models for, respectively, an n-Channel MOS field-effect transistor (MOSFET) and p-Channel MOSFET. These are advanced device models and no thorough description and motivation of the many input parameters are attempted here. Many device manufacturers provide model parameters for the MOSFET models. For any particular make of MOSFET, the device manufacturer should be the primary source of information.

NODE CONNECTIONS

Specify four **Node names** for the connection nodes for the **n-Channel MOSFET** or **p-Channel MOSFET** device. These represent the *drain*, *gate*, *source*, and *bulk* nodes, respectively.

MODEL PARAMETERS

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular MOSFET, the device manufacturer should be the primary source of information.

> The interested reader is referred to Ref. 1 for more details on semiconductor modeling within circuits.
>
> For an explanation of the **Model Parameters** see MOSFET Transistors.

Mutual Inductance

The **Mutual Inductance** allows specifying a coupling between two existing Inductor features in the circuit. The mutual inductance of the coupling is

$$M = k \sqrt{L_1 L_2}$$

where k is the coupling factor and L_1 and L_2 are the inductances of the inductors.

Enter values or expressions for the:

- **Coupling factor** k (dimensionless). The value must be between 0 and 1, and the default is 0.98.

- **First inductance** L_1 (SI unit: H) and **Second inductance** L_2 (SI unit: H). These must be set to two different Inductor features in the circuit.

Transformer

The **Transformer** feature represents either a combination of two Inductor and a Mutual Inductance features, or an ideal transformer.

NODE CONNECTIONS

Enter or edit the table in the **Node names** column for the **primary** and **secondary** node connections.

DEVICE PARAMETERS

Choose a **Transformer model** — **Specify inductors** (the default) or **Ideal transformer**.

For **Specify inductors** enter values or expressions for the:

- **Coupling factor** k (dimensionless). The default is 0.98.
- **First inductance** L_1 (SI unit: H). The default is 1 mH.
- **Second inductance** L_2 (SI unit: H). The default is 1 mH.

For **Ideal transformer** enter values or expressions for the **Winding ratio** N_1/N_2 (dimensionless). The default is 10.

Diode

The **Diode** device model ( ) is a large-signal model for a diode. It is an advanced device model and no thorough description and motivation of the many input parameters are attempted here. The interested reader is referred to Ref. 1 for more details on semiconductor modeling within circuits. Many device manufacturers

provide model parameters for this diode model. For any particular make of diode, the device manufacturer should be the primary source of information.

NODE CONNECTIONS

Specify two **Node names** for the positive and negative nodes for the **Diode** device.

MODEL PARAMETERS

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular diode, the device manufacturer should be the primary source of information.

🌕   For an explanation of the **Model Parameters** see Diode.

External I vs. U

The **External I vs. U** (▣) feature connects an arbitrary voltage measurement (for example, a circuit terminal or circuit port boundary or a coil domain from another physics interface) as a voltage source between two nodes in the electrical circuit. The resulting circuit current from the first node to the second node is typically coupled back as a prescribed current source in the context of the voltage measurement.

NODE CONNECTIONS

Specify the two **Node names** for the connecting nodes for the voltage source. The first node represents the positive reference terminal.

EXTERNAL DEVICE

Enter the source of the **Voltage**. If circuit or current excited terminals or circuit ports are defined on boundaries or domains or a multiturn coil domains is defined in other physics interfaces, these display as options in the **Voltage** list. Also select the **User defined** option and enter your own voltage variable, for example, using a suitable coupling operator. For inductive or electromagnetic wave propagation models, the voltage measurement must be performed as an integral of the electric field because the electric

potential only does not capture induced EMF. Also the integration must be performed over a distance that is short compared to the local wavelength.

| | Except when coupling to a circuit terminal, circuit port, or coil, the current flow variable must be manually coupled back in the electrical circuit to the context of the voltage measurement. This applies also when coupling to a current excited terminal. The name of this current variable follows the convention `cirn.IvsUm_i`, where `cirn` is the tag of the Electrical Circuit interface node and `IvsUm` is the tag of the **External I vs. U** node. The tags are typically displayed within curly brackets {} in the Model Builder. |

| | Nonlocal Couplings and Coupling Operators in the *COMSOL Multiphysics Reference Manual* |

External U vs. I

The **External U vs. I** (⊡) feature connects an arbitrary current measurement (for example, a coil domain from another physics interface) as a current source between two nodes in the electrical circuit. The resulting circuit voltage between the first node and the second node is typically coupled back as a prescribed voltage source in the context of the current measurement.

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.

EXTERNAL DEVICE
Enter the source of the **Current**. Voltage excited terminals or lumped ports defined on boundaries in other physics interfaces are natural candidates but do not appear as options in the **Voltage** list because those do not have an accurate built-in current

measurement variable. A **User defined** option must be selected and a current variable entered, for example, using a suitable coupling operator.

> **!** The voltage variable must be manually coupled back in the electrical circuit to the context of the current measurement. This applies also when coupling to a voltage excited terminal or lumped port. The name of this voltage variable follows the convention `cirn.UvsIm_v`, where `cirn` is the tag of the Electrical Circuit interface node and `UvsIm` is the tag of the **External U vs. I** node. The tags are typically displayed within curly brackets {} in the Model Builder.

> **Q** Nonlocal Couplings and Coupling Operators in the *COMSOL Multiphysics Reference Manual*

External I-Terminal

The **External I-Terminal** (⊕) feature connects an arbitrary voltage-to-ground measurement (for example, a circuit terminal from another physics interface) as a voltage-to-ground assignment to a node in the electrical circuit. The resulting circuit current from the node is typically coupled back as a prescribed current source in the context of the voltage measurement. This node does not apply when coupling to inductive or electromagnetic wave propagation models because then voltage must be defined as a line integral between two points rather than a single point measurement of electric potential. For such couplings, use the External I vs. U node instead.

NODE CONNECTIONS

Set the **Node name** for the connecting node for the voltage assignment.

EXTERNAL TERMINAL

Enter the source of the **Voltage**. If circuit- or current-excited terminals are defined on boundaries in other physics interfaces, these display as options in the **Voltage** list. Also

select the **User defined** option and enter a voltage variable, for example, using a suitable coupling operator.

	• Except when coupling to a circuit terminal, the current flow variable must be manually coupled back in the electrical circuit to the context of the voltage measurement. This applies also when coupling to a current excited terminal. The name of this current variable follows the convention `cirn.termIm_i`, where `cirn` is the tag of the Electrical Circuit interface node and `termIm` is the tag of the **External I-Terminal** node. The tags are typically displayed within curly brackets `{}` in the Model Builder.
	• When connecting the finite element model between two circuit nodes that both are not grounded, the **External I-Terminal** node cannot be used. In this case, use External I vs. U or External U vs. I instead.

	Nonlocal Couplings and Coupling Operators in the *COMSOL Multiphysics Reference Manual*.

SPICE Circuit Import

Right-click the **Electrical Circuit** ( ) feature node to import an existing SPICE netlist (select **Import Spice Netlist**). A window opens — enter a file location or browse your directories to find one. The default file extension for a SPICE netlist is `.cir`. The SPICE circuit import translates the imported netlist into Electrical Circuit interface nodes so these define the subset of SPICE features that can be imported.

	See SPICE Import and Export about the supported SPICE commands.

SPICE Circuit Export

Right-click the **Electrical Circuit** ( ) feature node to export the current circuit to the SPICE netlist file format (select **Export Spice Netlist**). A window opens — enter a

file location or browse your directories to find one. The default file extension for a SPICE netlist is `.cir`. The compatible circuit nodes are exported as SPICE devices

See for more details on the supported SPICE commands.

Theory for the Electrostatics Interface

The Electrostatics Interface is available for 3D, 2D in-plane, and 2D axisymmetric components. Applications with electrostatic equations include high-voltage apparatus, electronic devices, and capacitors. The term "statics" is not to be interpreted literally — it is the observation time, or time scale at which the applied excitation changes, that is short compared to the charge relaxation time; also, the electromagnetic wavelength and skin depth are very large compared to the size of the domain of interest.

If you do not know whether to use the Electric Currents or the Electrostatics interface, which both solve for the scalar electric potential V, consider using an explicit charge transport model. See Charge Relaxation Theory.

- Debye Dispersion

Charge Relaxation Theory

COMSOL Multiphysics includes physics interfaces for the modeling of static electric fields and currents. Deciding what specific physics interface and study type to select for a particular modeling situation requires a basic understanding of the charge dynamics in conductors.

The different physics interfaces involving only the scalar electric potential can be interpreted in terms of the charge relaxation process. The fundamental equations involved are *Ohm's law* for the conduction current density

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

the *equation of continuity*

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

and *Gauss' law*

$$\nabla \cdot (\varepsilon \mathbf{E}) = \rho$$

By combining these, one can deduce the following differential equation for the space charge density in a homogeneous medium

$$\frac{\partial \rho}{\partial t} + \frac{\sigma}{\varepsilon}\rho = 0$$

This equation has the solution

$$\rho(t) = \rho_0 e^{-t/\tau}$$

where

$$\tau = \frac{\varepsilon}{\sigma}$$

is called the charge relaxation time. For a good conductor like copper, τ is of the order of 10^{-19} s, whereas for a good insulator like silica glass, it is of the order of 10^3 s. For a pure insulator, it becomes infinite.

When modeling real-world devices, there is not only the *intrinsic time scale* of the charge relaxation time but also an *external time scale* t at which a device is energized or the observation time. It is the relation between the external time scale and the charge relaxation time that determines what physics interface and study type to use. The results are summarized in Table 3-3 below,

TABLE 3-3: SUITABLE PHYSICS INTERFACE AND STUDY TYPE FOR DIFFERENT TIME-SCALE REGIMES.

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

FIRST CASE: $\tau \gg T$

If the external time scale is short compared to the charge relaxation time, the charges do not have time to redistribute to any significant degree. Thus the charge distribution can be considered as a given model input. The best approach is to solve the Electrostatics formulation using the electric potential V.

By combining the definition of the potential with Gauss' law, you can derive the classical Poisson's equation. Under static conditions, the electric potential V is defined by the equivalence $\mathbf{E} = -\nabla V$. Using this together with the constitutive relation $\mathbf{D} = \varepsilon_0\mathbf{E} + \mathbf{P}$ between $\mathbf{D}$ and $\mathbf{E}$, you can rewrite Gauss' law as a variant of Poisson's equation

$$-\nabla \cdot (\varepsilon_0 \nabla V - \mathbf{P}) = \rho$$

This equation is used in the Electrostatics interface. It is worth noting that Gauss' law does not require the charge distribution to be static. Thus, provided dynamics are slow enough that induced electric fields can be neglected and hence a scalar electric potential is justified, the formulation can be used also in the Time Dependent study type. That typically involves either prescribing the charge dynamics or coupling a separate formulation for this.

Such separate charge transport formulations can be found in the Plasma Module, the Semiconductor Module, and the Chemical Reaction Engineering Module.

SECOND CASE: $\tau << T$

If the external time scale is long compared to the charge relaxation time, the stationary solution to the equation of continuity has been reached. In a stationary coordinate system, a slightly more general form of Ohm's law than above states that

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

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

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

To handle current sources, the equation can be generalized to

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

This equation is used in the static study type for the Electric Currents interface.

GENERAL CASE: CHARGE DYNAMICS

If the charge relaxation time is comparable to the external time scale, the Time Dependent or Frequency Domain study types for the Electric Currents interface must be used.

Combining the time-harmonic equation of continuity

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

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

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

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

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

and the resulting equation becomes

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

These dynamic formulations are valid as long as induced electric fields can be ignored and hence the electric field is essentially curl free. This condition is fulfilled provided that skin effect and wave propagation effects can be ignored. The skin depth must be much larger than the geometrical dimensions of the modeled device and so must the wavelength. Note also that these formulations can be used to model dielectric regions of capacitive/resistive devices even though the interior of electrodes sometimes does not meet the large skin depth condition. In that case, the electrodes must only be represented as boundary conditions (fixed or floating potential). The interior, metallic domains are not included in the analysis. Obviously, this is only a valid approach for devices where metallic electrodes do not entirely bypass (short circuit) the capacitive/resistive layers. If metallic electrodes short circuit the capacitive/resistive layers, the time evolution of the current is determined by inductive and resistive effects with very little influence from the capacitive layers. Then the Magnetic Fields interface is the appropriate modeling tool.

Electrostatics Equations

Under static conditions, the electric potential, V, is defined by the relationship:

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

Combining this equation with the constitutive relationship $\mathbf{D} = \varepsilon_0\mathbf{E} + \mathbf{P}$ between the electric displacement $\mathbf{D}$ and the electric field $\mathbf{E}$, it is possible to represent Gauss' law as the following equation:

$$-\nabla \cdot (\varepsilon_0\nabla V - \mathbf{P}) = \rho$$

In this equation, the physical constant, ε_0 (SI unit: F/m) is the permittivity of vacuum, $\mathbf{P}$ (SI unit: C/m^2) is the electric polarization vector, and ρ (SI unit: C/m^3) is a space charge density. This equation describes the electrostatic field in dielectric materials.

For in-plane 2D modeling, the Electrostatics interface assumes a symmetry where the electric potential varies only in the x and y directions and is constant in the z direction. This implies that the electric field, $\mathbf{E}$, is tangential to the xy-plane. With this symmetry, the same equation is solved as in the 3D case. The physics interface solves the following equation where d is the thickness in the z direction:

$$-\nabla \cdot d(\varepsilon_0 \nabla V - \mathbf{P}) = \rho$$

The axisymmetric version of the physics interface considers the situation where the fields and geometry are axially symmetric. In this case, the electric potential is constant in the φ direction, which implies that the electric field is tangential to the rz-plane.

The Electrostatics Interface in Time Dependent or Frequency Domain Studies

The Electrostatics Interface can also be solved in a dynamic study (Time Dependent or Frequency Domain). The equation system solved, however, is typically always the one presented in the previous section for the stationary case, in which no transient electromagnetic effects are taken into account. The difference is that the sources of the problem (charge densities, electric potential) are assumed to be time-varying (in a Time Dependent study) or time-harmonic (in a Frequency Domain study).

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

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

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

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

Debye Dispersion

Ampere's law is written as

$$\nabla \times \mathbf{H} = \mathbf{J} \tag{3-1}$$

The total current is given by

$$\mathbf{J} = \mathbf{J}_i + \frac{\partial \mathbf{D}}{\partial t} + \mathbf{J}_p \tag{3-2}$$

where

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

is the induction current,

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

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

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

$$\mathbf{J}_p = \sum_{m=1}^{N} \mathbf{J}_m$$

where each term obeys an equation of the following form:

$$\tau_m \frac{\partial \mathbf{J}_m}{\partial t} + \mathbf{J}_m = \varepsilon_0 \Delta \varepsilon_m \frac{\partial \mathbf{E}}{\partial t} \tag{3-3}$$

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

Introduce new variables, auxiliary electric field from

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

Equation 3-3 can be integrated once with respect to time assuming zero initial fields

$$\tau_m \frac{\partial \mathbf{e}_m}{\partial t} + \mathbf{e}_m = \mathbf{E} \tag{3-4}$$

The polarization current can be expressed the as either

$$\mathbf{J}_p = \frac{\partial}{\partial t} \sum_{m=1}^{N} \varepsilon_0 \Delta\varepsilon_m \mathbf{e}_m \qquad (3\text{-}5)$$

or

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

By applying the divergence operator, Equation 3-1 yields the current conservation equation:

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

Assume that the electric conductivity is small, so that the induction current can be neglected. Using Equation 3-5, the current conservation equation can be written as

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

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

$$\nabla \cdot \left(\varepsilon_0 \varepsilon_\infty \mathbf{E} + \sum_{m=1}^{N} \varepsilon_0 \Delta\varepsilon_m \mathbf{e}_m \right) = 0 \qquad (3\text{-}6)$$

Equation 3-6 can be solved using the scalar electric potential V as a dependent variable determine the electric field as $\mathbf{E} = -\nabla V$, and it needs to be solved together with N vector Equation 3-4 for N vector dependent variables $\mathbf{e}_m$. In time domain, these auxiliary field variables can be treated as the local state variables, and each of N corresponding ODE equations can be integrated locally.

FREQUENCY DOMAIN

In frequency domain, Equation 3-5 for the auxiliary variables can be solved analytically, so that Equation 3-6 gives

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

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

$$\nabla \cdot (\varepsilon_0 (\varepsilon' - j\varepsilon'') \mathbf{E}) = 0$$

where

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

and

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

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

EIGENFREQUENCY STUDY

Direct use of the complex frequency-dependent permittivity would lead to a nonlinear eigenvalue problem, for which the solution would depend on the eigenvalue linearization point (transform point frequency, specified on the solver). The solution would be accurate only for the eigenfrequency that is close to the transform point frequency. Thus, the eigenfrequencies would need to be computed one by one using iterative updates of the transform point frequency value.

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

$$j\omega\tau_m \mathbf{e}_m + \mathbf{e}_m = \mathbf{E} \tag{3-7}$$

together with Equation 3-7. Even though this approach requires extra degrees of freedom, it will produce frequency independent contributions to the damping and stiffness matrices. The corresponding linear damped eigenvalue problem can be solved

using the default solver in a single run for any specified number of eigenfrequencies which will be computed exactly.

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

CONSTANT LOSS TANGENT

In frequency domain, the loss tangent is defined as

$$\eta = \tan\delta = \varepsilon''/\varepsilon' \tag{3-8}$$

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

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

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

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

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

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

The corresponding relative permittivity contribution is given by

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

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

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

The accuracy can be selected as normal to give

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

for

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

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

Alternatively, the accuracy can be selected as high to get

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

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

Thus computed parameters will be used automatically in a Time Dependent study using Equation 3-4, and in an Eigenfrequency study using Equation 3-8.

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

TEMPERATURE EFFECTS

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

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

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

The implication is that the problem can be solved using the original material data, provided that the time is transformed into the reduced time. Think of the shift function $\alpha_T(T)$ as a multiplier to the relaxation time in Equation 3-4 and Equation 3-8.

Vogel-Fulcher Shift
The shift function is given by

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

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

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

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

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

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

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

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

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

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

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

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

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

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

here, a base-e logarithm is used, Q is the activation energy (SI unit: J/mol), R is the universal gas constant, T is the current temperature, T_{ref} is a reference temperature, χ is a dimensionless activation energy fraction, and T_f is the so-called fictive temperature. The fictive temperature is given as the weighted average of partial fictive temperatures.

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

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

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

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

REFERENCES FOR DEBYE DISPERSION
1. N.S. Stoykov, T.A. Kuiken, M.M. Lowery, and A. Taflove, "Finite-element time-domain algorithms for modeling linear Debye and Lorentz dielectric dispersions

at low frequencies," *IEEE Transactions on Biomedical Engineering*, vol. 50, no. 9, pp. 1100–1107, 2003.

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

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

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

Theory for the Electric Currents Interface

The Electric Currents Interface solves a current conservation problem for the scalar electric potential V and is available for 3D, 2D in-plane, and 2D axisymmetric components. Electrolysis, conductors and the computation of resistances of grounding plates are examples that involve conductive media with electrical conductivities and electric currents. If you are uncertain of whether to use the Electric Currents or the Electrostatics interface, which both solve for the scalar electric potential V, refer to the section on Charge Relaxation Theory.

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

In this section:

- Electric Currents Equations in Steady State
- Dynamic Electric Currents Equations

Electric Currents Equations in Steady State

When handling stationary electric currents in conductive media you must consider the stationary equation of continuity. In a stationary coordinate system, the point form of *Ohm's law* states that:

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

where σ is the electrical conductivity (SI unit: S/m), and $\mathbf{J}_e$ is an externally generated current density (SI unit: A/m^2). The static form of the equation of continuity then states:

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

To handle current sources, you can generalize the equation to:

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

In planar 2D the Electric Currents interface assumes that the model has a symmetry where the electric potential varies only in the x and y directions and is constant in the z direction. This implies that the electric field, $\mathbf{E}$, is tangential to the xy-plane. The Electric Currents interface then solves the following equation, where d is the thickness in the z direction:

$$-\nabla \cdot d(\sigma \nabla V - \mathbf{J}_e) = dQ_j \qquad (3\text{-}9)$$

In 2D axisymmetry, the Electric Currents interface considers the situation where the fields and geometry are axially symmetric. In this case, the electric potential is constant in the φ direction, which implies that the electric field is tangential to the rz-plane.

Dynamic Electric Currents Equations

In the Frequency Domain and Time Dependent study types dynamic formulations accounting for both conduction currents and displacement currents are used.

Combining the time-harmonic equation of continuity

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

with the equation

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

and generalized to handle current sources yields the following equation:

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

For the transient case, using the transient equation of continuity

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

and generalized to handle current sources the resulting equation becomes:

$$-\nabla \cdot \frac{\partial}{\partial t}(\varepsilon_0 \nabla V - \mathbf{P}) - \nabla \cdot (\sigma \nabla V - \mathbf{J}_e) \ = \ Q_j \qquad (3\text{-}10)$$

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

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

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

Theory for the Electrical Circuit Interface

The Electrical Circuit Interface theory is discussed in this section:

• Electric Circuit Modeling and the Semiconductor Device Models

• Bipolar Transistors

• MOSFET Transistors

• Diode

• Reference for the Electrical Circuit Interface

   🔍 | Connecting to Electrical Circuits

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.

• The field model is used to get a better, more accurate description of a single device in the electrical circuit model.

• The electrical circuit is used to drive or terminate the device in the field model in such a way that it makes more sense to simulate both as a tightly coupled system.

The Electrical Circuit interface makes it possible to add nodes representing circuit elements directly to the Model Builder tree in a COMSOL Multiphysics model. The circuit variables can then be connected to a physical device model to perform co-simulations of circuits and multiphysics. The model acts as a device connected to the circuit so that its behavior is analyzed in larger systems.

The fundamental equations solved by the Electrical Circuit interface are Kirchhoff's circuit laws, which in turn can be deduced from Maxwell's equations. The supported study types are Stationary, Frequency Domain, and Time Dependent.

There are three more advanced large-signal semiconductor device features available in the Electrical Circuit interface. The equivalent circuits and the equations defining their non-ideal circuit elements are described in this section. For a more complete treatise on semiconductor device modeling see Ref. 1.

Bipolar Transistors

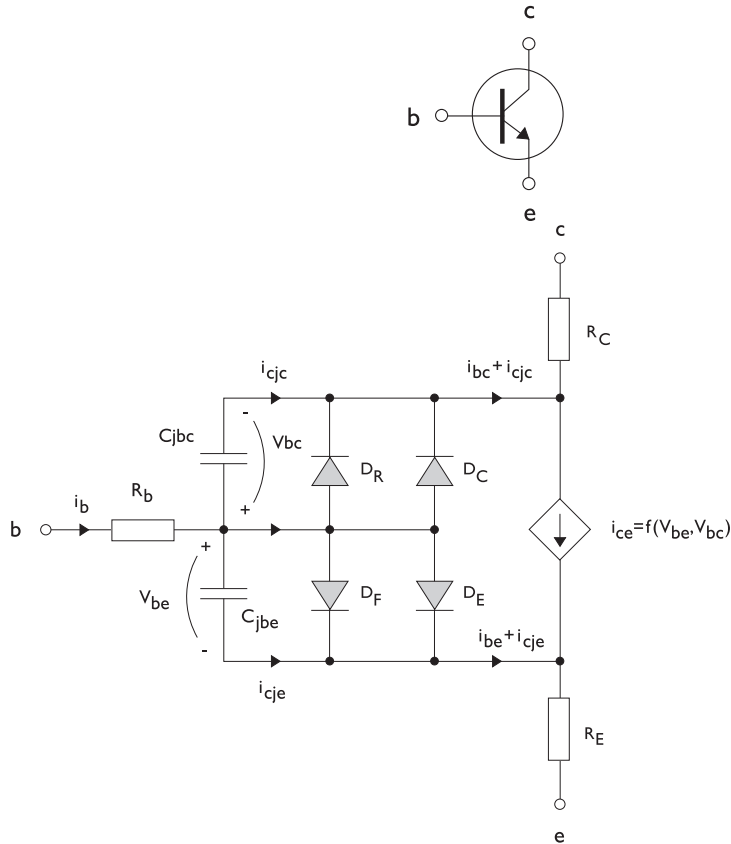Figure 3-1 illustrates the equivalent circuit for the npn bipolar junction transistor.

Figure 3-1: A circuit for the bipolar transistor.

The pnp transistor model is similar in all regards to the npn transistor, with the difference that the polarities of the currents and voltages involved are reversed. The following equations are used to compute the relations between currents and voltages in the circuit.

$$v_{rb} = \frac{1}{A}\left(R_{BM} - \frac{R_B - R_{BM}}{f_{bq}}\right)i_b$$

$$f_{bq} = \frac{1}{2\left(1 - \dfrac{v_{bc}}{V_{AF}} - \dfrac{v_{be}}{V_{AR}}\right)}\left(1 + \sqrt{1 + 4I_S\left(\frac{e^{\frac{v_{be}}{N_F V_T}} - 1}{I_{KF}A} + \frac{e^{\frac{v_{bc}}{N_R V_T}} - 1}{I_{KR}A}\right)}\right)$$

$$i_{be} = A\left(\frac{I_S}{B_F}\left(e^{\frac{v_{be}}{N_F V_T}} - 1\right) + I_{SE}\left(e^{\frac{v_{be}}{N_E V_T}} - 1\right)\right)$$

$$i_{bc} = A\left(\frac{I_S}{B_R}\left(e^{\frac{v_{bc}}{N_R V_T}} - 1\right) + I_{SC}\left(e^{\frac{v_{bc}}{N_C V_T}} - 1\right)\right)$$

$$i_{ce} = A\left(\frac{I_S}{f_{bq}}\left(e^{\frac{v_{be}}{N_F V_T}} + e^{\frac{v_{bc}}{N_C V_T}}\right)\right)$$

$$V_T = \frac{k_B T_{NOM}}{q}$$

There are also two capacitances that use the same formula as the junction capacitance of the diode model. In the parameter names below, replace x with C for the base-collector capacitance and E for the base-emitter capacitance.

$$C_{jbx} = AC_{Jx} \times \begin{cases} \left(1 - \dfrac{v_{bx}}{V_{Jx}}\right)^{-M_{Jx}} & v_{bx} < F_C V_{Jx} \\ (1 - F_C)^{-1 - M_{Jx}}\left(1 - F_C(1 + M_{Jx}) + M_{Jx}\dfrac{v_{bx}}{V_{Jx}}\right) & v_{bx} \geq F_C V_{Jx} \end{cases}$$

The model parameters are listed in the table below.

TABLE 3-1: BIPOLAR TRANSISTOR MODEL PARAMETERS.

PARAMETER	DEFAULT	DESCRIPTION
B_F	100	Ideal forward current gain
B_R	1	Ideal reverse current gain
C_{JC}	0 F/m^2	Base-collector zero-bias depletion capacitance
C_{JE}	0 F/m^2	Base-emitter zero-bias depletion capacitance
F_C	0.5	Breakdown current
I_{KF}	Inf (A/m^2)	Corner for forward high-current roll-off

TABLE 3-1: BIPOLAR TRANSISTOR MODEL PARAMETERS.

PARAMETER	DEFAULT	DESCRIPTION
I_{KR}	Inf (A/m^2)	Corner for reverse high-current roll-off
I_{S}	10^{-15} A/m^2	Saturation current
I_{SC}	0 A/m^2	Base-collector leakage saturation current
I_{SE}	0 A/m^2	Base-emitter leakage saturation current
M_{JC}	1/3	Base-collector grading coefficient
M_{JE}	1/3	Base-emitter grading coefficient
N_C	2	Base-collector ideality factor
N_E	1.4	Base-emitter ideality factor
N_F	1	Forward ideality factor
N_R	1	Reverse ideality factor
R_B	0 $\Omega \cdot$m^2	Base resistance
R_{BM}	0 $\Omega \cdot$m^2	Minimum base resistance
R_C	0 $\Omega \cdot$m^2	Collector resistance
R_E	0 $\Omega \cdot$m^2	Emitter resistance
T_{NOM}	298.15 K	Device temperature
V_{AF}	Inf (V)	Forward Early voltage
V_{AR}	Inf (V)	Reverse Early voltage
V_{JC}	0.71 V	Base-collector built-in potential
V_{JE}	0.71 V	Base-emitter built-in potential

MOSFET Transistors

Figure 3-2 illustrates an equivalent circuit for the n-channel MOSFET transistor. The p-channel MOSFET transistor is treated similarly, but the polarities of the involved voltages are reversed.
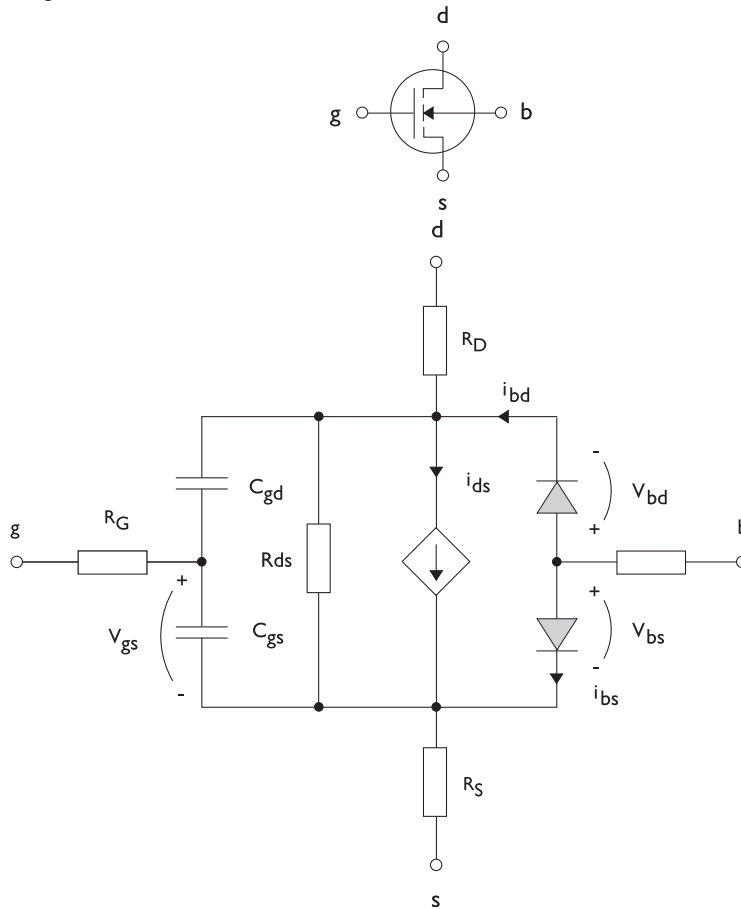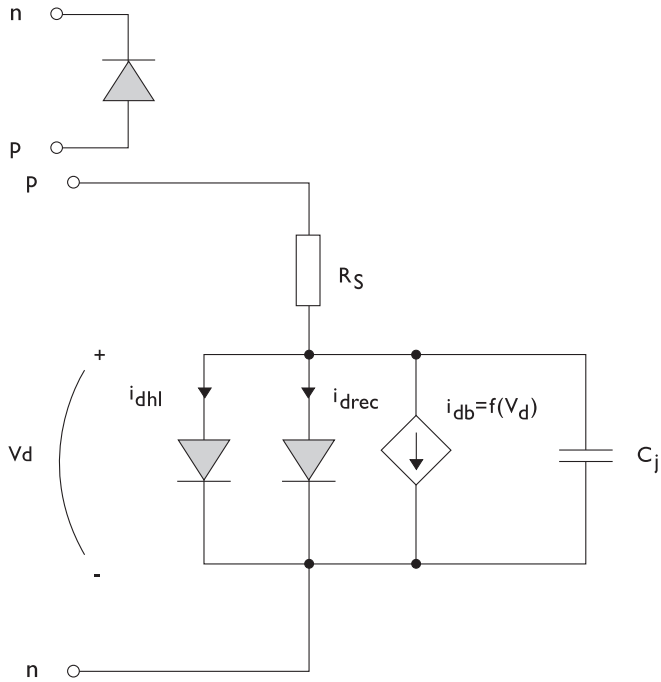

Figure 3-2: A circuit for the MOSFET transistor.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$i_{ds} = \begin{cases} \dfrac{W}{L}\dfrac{K_P}{2}(1 + \Lambda v_{ds})v_{ds}(2v_{th} - v_{ds}) & v_{ds} < v_{th} \\[3mm] \dfrac{W}{L}\dfrac{K_P}{2}(1 + \Lambda v_{ds})v_{th}^2 & v_{ds} \geq v_{th} \\[3mm] 0 & v_{ds} < v_{th} \leq 0 \end{cases}$$

$$v_{th} = v_{gs} - (V_{TO} + \Gamma(\sqrt{\Phi - v_{bs}} - \sqrt{\Phi}))$$

$$i_{bd} = I_S\left(e^{\frac{v_{bd}}{NV_T}} - 1\right)$$

$$i_{bs} = I_S\left(e^{\frac{v_{bs}}{NV_T}} - 1\right)$$

$$V_T = \frac{k_B T_{NOM}}{q}$$

There are also several capacitances between the terminals

$$C_{gd} = C_{gd0}W$$
$$C_{gs} = C_{gs0}W$$

$$C_{jbd} = C_{BD} \times \begin{cases} \left(1 - \dfrac{v_{bd}}{P_B}\right)^{-M_J} & v_{bx} < F_C P_B \\[3mm] (1 - F_C)^{-1 - M_J}\left(1 - F_C(1 + M_J) + M_J\dfrac{v_{bx}}{P_B}\right) & v_{bx} \geq F_C P_B \end{cases}$$

The model parameters are as follows:

TABLE 3-2: MOSFET TRANSISTOR MODEL PARAMETERS.

PARAMETER	DEFAULT	DESCRIPTION
C_{BD}	0 F/m	Bulk-drain zero-bias capacitance
C_{GDO}	0 F/m	Gate-drain overlap capacitance
C_{GSO}	0 F/m	Gate-source overlap capacitance
F_C	0.5	Capacitance factor
I_S	1e-13 A	Bulk junction saturation current
K_P	2e-5 A/V^2	Transconductance parameter
L	50e-6 m	Gate length
M_J	0.5	Bulk junction grading coefficient

TABLE 3-2: MOSFET TRANSISTOR MODEL PARAMETERS.

PARAMETER	DEFAULT	DESCRIPTION
N	1	Bulk junction ideality factor
P_B	0.75 V	Bulk junction potential
R_B	0 Ω	Bulk resistance
R_D	0 Ω	Drain resistance
R_{DS}	Inf (Ω)	Drain-source resistance
R_G	0 Ω	Gate resistance
R_S	0 Ω	Source resistance
T_{NOM}	298.15 K	Device temperature
V_{TO}	0 V	Zero-bias threshold voltage
W	50e-6 m	Gate width
Γ (GAMMA)	1 $V^{0.5}$	Bulk threshold parameter
Φ (PHI)	0.5 V	Surface potential
Λ (LAMBDA)	0 1/V	Channel-length modulation

Diode

Figure 3-3 illustrates equivalent circuit for the diode.

Figure 3-3: A circuit for the diode.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$i_d = i_{dhl} + i_{drec} + i_{db} + i_c$$

$$i_{dhl} = I_S\left(e^{\frac{v_d}{NV_T}} - 1\right)\frac{1}{\sqrt{1 + \dfrac{I_S}{I_{KF}}\left(e^{\frac{v_d}{NV_T}} - 1\right)}}$$

$$i_{drec} = I_{SR}\left(e^{\frac{v_d}{N_R V_T}} - 1\right)$$

$$i_{db} = I_{BV}e^{-\frac{v_d + B_V}{N_{BV}V_T}}$$

$$C_j = C_{J0} \times \begin{cases} \left(1 - \dfrac{v_d}{V_J}\right)^{-M} & v_d < F_C V_J \\[2ex] (1 - F_C)^{-1-M}\left(1 - F_C(1 + M) + M\dfrac{v_d}{V_J}\right) & v_d \geq F_C V_J \end{cases}$$

$$V_T = \frac{k_B T_{NOM}}{q}$$

where the following model parameters are required.

TABLE 3-3: DIODE TRANSISTOR MODEL PARAMETERS.

PARAMETER	DEFAULT	DESCRIPTION
B_V	Inf (V)	Reverse breakdown voltage
C_{J0}	0 F	Zero-bias junction capacitance
F_C	0.5	Forward-bias capacitance coefficient
I_{BV}	1e-09 A	Current at breakdown voltage
I_{KF}	Inf (A)	Corner for high-current roll-off
I_S	1e-13 A	Saturation current
M	0.5	Grading coefficient
N	1	Ideality factor
N_{BV}	1	Breakdown ideality factor
N_R	2	Recombination ideality factor
R_S	0 Ω	Series resistance
T_{NOM}	298.15 K	Device temperature
V_J	1.0 V	Junction potential

Reference for the Electrical Circuit Interface

1. P. Antognetti and G. Massobrio, *Semiconductor Device Modeling with Spice*, 2nd ed., McGraw-Hill, 1993.

4

Fluid Flow Interfaces

This chapter describes the physics interfaces found under the **Fluid Flow** branch (≋).

The documentation of fluid structure interaction is found in the Structural Mechanics Module documentation, and the documentation for thin-film flow in the CFD Module documentation.

The list below details the documentation for the MEMS Module physics interfaces on the **Fluid Flow** branch:

- The Thin-Film Flow Interfaces (in the *CFD Module User's Guide*)
- Theory for the Thin-Film Flow Interfaces (in the *CFD Module User's Guide*)
- The Fluid-Solid Interaction Interface (in the *Structural Mechanics Module User's Guide*)
- The Solid Thin-Film Damping Interface (in the *Structural Mechanics Module User's Guide*)
- The Piezoelectricity Interface, Solid (in the *Structural Mechanics Module User's Guide*)

See The Single-Phase Flow, Laminar Flow Interface in the *COMSOL Multiphysics Reference Manual* for other Fluid Flow interface and feature node settings.

5

Structural Mechanics Interfaces

The MEMS module includes many of the features found in the Solid Mechanics interface in the Structural Mechanics Module.

The documentation of shared features is found in the Structural Mechanics Module documentation. Where indicated in the list below, links to the *Structural Mechanics Module User's Guide* are included.

The list below details the documentation for the MEMS Module physics interfaces on the structural mechanics, acoustics, and fluid flow branches in the **Add Physics** dialog:

- The Electromechanics Interface
- Theory for the Electromechanics Interface
- The Electromechanics, Boundary Elements Interface
- The Pyroelectricity Interface
- The Thermoelasticity Interface
- Theory for the Thermoelasticity Interface
- The Solid Mechanics Interface (in the *Structural Mechanics Module User's Guide*)

- The Elastic Waves, Time Explicit Interface (in the Elastic Waves Interfaces chapter of the *Acoustics Module User's Guide*)

- The Thermal Stress, Solid Interface (in the Multiphysics Interfaces and Couplings chapter of the *Structural Mechanics Module User's Guide*)

- The Piezoelectricity Interface, Solid (in the Multiphysics Interfaces and Couplings chapter of the *Structural Mechanics Module User's Guide*)

- The Electrostriction Interface (in the Multiphysics Interfaces and Couplings chapter of the *Structural Mechanics Module User's Guide*)

- The Ferroelectroelasticity Interface (in the Multiphysics Interfaces and Couplings chapter of the *Structural Mechanics Module User's Guide*)

- The Fluid-Solid Interaction Interface (in the Multiphysics Interfaces and Couplings chapter of the *Structural Mechanics Module User's Guide*)

- The Fluid-Solid Interaction, Fixed Geometry Interface (in the Multiphysics Interfaces and Couplings chapter of the *Structural Mechanics Module User's Guide*)

- The Fluid-Solid Interaction, Conjugate Heat Transfer Interface (in the Multiphysics Interfaces and Couplings chapter of the *Structural Mechanics Module User's Guide*)

- Hygroscopic Swelling Coupling (in the Multiphysics Interfaces and Couplings chapter of the *Structural Mechanics Module User's Guide*)

The Electromechanics Interface

The **Electromechanics** interface (⬥), found under the **Electromagnetics-Structure Interaction** (⬥) branch within the **Structural Mechanics** branch (⬥) or the **Electromagnetics and Mechanics** (⬥) branch within the **AC/DC** branch (⬥) when adding a physics interface, combines Solid Mechanics and Electrostatics with a moving mesh functionality to model the deformation of electrostatically actuated structures. The **Electromechanics** interface is available for 3D, planar 2D, and axisymmetric 2D geometries.

When this multiphysics interface is added using the **Model Wizard**, a **Solid Mechanics** interface and an **Electrostatics** interfaces are added to the **Model Builder**. In addition, the **Multiphysics Couplings** node is added, which automatically includes the multiphysics coupling feature **Electromechanical Forces**.

	In previous versions of COMSOL Multiphysics (up to version 5.3), a specific physics interface called **Electromechanics** was added to the **Model Builder**. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see The Multiphysics Branch and Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

The full functionality of the Solid Mechanics and Electrostatics interfaces is accessible under the corresponding interfaces at the domain, boundary line, or point level in the geometry. The participating **Electrostatics** interface by default adds the **Charge Conservation** node with its selection set to all domains, and the material type set to Solid. This node serves to represent solid dielectric material domains. Any number of **Charge Conservation** nodes can be also added manually to the interface and configured to represent either solid or nonsolid (for example, air or free space) materials.

Electromechanical Forces

Use **Electromechanics Forces** (⬉) coupling to model electromechanical interaction between deformable solids and electric fields, particularly in case when the deformations of the structure can significantly affect the electric field distribution.

	The presence of such multiphysics node in the model will enforce the **Include geometric nonlinearity** option on all applicable study nodes.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is eme1.

DOMAIN SELECTION

The domain selection is locked, so that all applicable domains are selected automatically. Such domains represent an intersection of the applicable domains under the corresponding **Electrostatics** and **Solid Mechanics** interfaces selected in the coupling feature.

In **Electrostatics** interface, the following two domains are applicable:

- **Charge Conservation**, if its material type input is set to **Solid**. Use this domain feature for solid dielectric materials.

- **Charge Conservation, Ferroelectric**

- **Terminal**. You can use the domain terminal feature to represent conductive solid materials such as, for example, doped silicon.

In **Solid Mechanics** interface, the following domain material feature are applicable:

- **Linear Elastic Material**

- **Nonlinear Elastic Material** (with the Nonlinear Structural Materials Module)

- **Hyperelastic Material** (with the Nonlinear Structural Materials Module).

	For nonsolid dielectric domains, remove them from the **Solid Mechanics** interface selection but keep them selected in the **Electrostatics** interface selection.

ELECTROMAGNETIC STRESS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

Here, you can select the **Electromagnetic Stress Tensor** to be used in the coupling. The following options are available:

- **Minkowsky** (default)
- **Einstein-Laub**
- **Chu**

See About different stress forms chapter for more details.

COUPLED INTERFACES

This section defines the physics interfaces involved in this Electromechanical Forces coupling. The **Solid mechanics** and **Electrostatics** lists include all applicable physics interfaces.

The default values depend on how the node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.

- If it is added automatically when the physics interface is chosen in the **Model Wizard** or **Add Physics** window, then the two participating physics interfaces are selected.

You can also select **None** from either list to uncouple the coupling from a physics interface. If the physics interface is removed from the **Model Builder**, for example **Solid**

Mechanics is deleted, then the **Solid mechanics** list defaults to **None** as there is nothing to couple to.

	If a physics interface is deleted and then added to the model again, then in order to reestablish the coupling, you need to choose the physics interface again from the lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present physics interface. See Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

Deforming Domain

When **Electromechanics** multiphysics interface is added using the **Model Wizard**, a **Deforming Domain** and a **Symmetry** nodes are added automatically in **Model Builder** under the **Definitions** node. These nodes serve to represent domains and boundaries where the mesh can deform. By default, both nodes have empty selection. Under the **Deforming Domain**, you can select those domain that represent nonsolid dielectric materials such as air, fluid, and free space. However, this is only needed if the boundaries of such domains experience significant changes due to the deformation or rotation of the adjacent solid domains. Otherwise, the moving mesh computations could just introduce unnecessary overhead and nonlinearity in the model (see Deformed Mesh Fundamentals). Domains selected in the Solid Mechanics interface are not applicable.

Under the **Deforming Domain**, you can also choose the **Mesh smoothing type**, by default set to **Hyperelastic**. More information of the smoothing type can be found in the Deforming Domain chapter in the *COMSOL Multiphysics Reference Manual*.

By default, the mesh is fixed at all external boundaries of the geometry. Use the **Symmetry** node to select the external boundaries, where the mesh can slide along the boundary. You can also add other types of boundary conditions for the mesh motion, for details see Moving Mesh Features in the *COMSOL Multiphysics Reference Manual*.

Theory for the Electromechanics Interface

The Electromechanics Interface theory is described in this section:

- Overview of Forces in Continuum Mechanics
- The Electromagnetic Stress Tensor
- References for the Electromechanics Interface

Overview of Forces in Continuum Mechanics

The deformation of linear elastic materials subject to external loads is described by Cauchy's equation:

$$\rho\frac{d^2\mathbf{u}}{dt^2} = \nabla \cdot \sigma + \mathbf{f}_v$$

where ρ is the density, $\mathbf{u}$ denotes the structural displacement vector, σ is the stress tensor, and $\mathbf{f}_v$ is an external volume force (such as gravity).

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

$$\mathbf{0} = \nabla \cdot \sigma + \mathbf{f}_v$$

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

$$(\sigma_2 - \sigma_1)\mathbf{n} = \mathbf{0} \tag{5-1}$$

where σ_1 and σ_2 represent the stress tensor in Materials 1 and 2, respectively, and $\mathbf{n}_1$ is the normal vector pointing out from the domain containing Material 1.
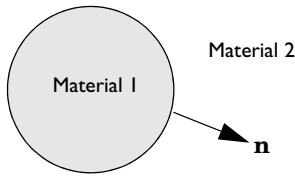
> For more information on the mechanical stress tensor for elastic materials, see the documentation for the physics interfaces. For example, Structural Mechanics Interfaces.

The Electromagnetic Stress Tensor

Within a vacuum or other medium, forces between charged bodies can be computed on the assumption that a fictitious state of stress exists within the field. Historically the consistency of this approach led 19th-century physicists to postulate the existence of the ether, a ubiquitous medium through which electromagnetic forces propagate. While these ideas have been superseded by the development of special relativity, the use of an electromagnetic stress tensor (also known as the Maxwell stress tensor) remains an accurate and convenient technique to compute electromagnetic forces.

Maxwell stresses exists in all dielectric materials, including air and even free space. However, the force magnitude is rather small, and it usually can cause significant deformations only at small dimension.

The electromagnetic stress tensor in a vacuum (in the absence of magnetic fields) is given by (Ref. 1):

$$\sigma_{\mathrm{EM, V}} = \varepsilon_0 \mathbf{E} \otimes \mathbf{E} - \frac{1}{2}(\varepsilon_0 \mathbf{E} \cdot \mathbf{E})I \tag{5-2}$$

Where $\mathbf{E}$ is the electric field, I is the identity tensor, ε_0 is the permittivity of free space, and

$$(\mathbf{E} \otimes \mathbf{E})_{ij} = E_i E_j$$

Within an isotropic linear dielectric under small deformations, the electric displacement vector is related to the electric field as (Ref. 2):

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

where the electric susceptibility χ can be a function of the mechanical strain in the material

$$\chi = \chi_0(I - 2\mathrm{T} + \mathrm{tr}(\mathrm{T})I)$$

where the small strain tensor is given by

$$\mathrm{T} = \frac{1}{2}(\nabla\mathbf{u} + \nabla\mathbf{u}^T)$$

The corresponding electromagnetic tress tensor takes the following form:

$$\sigma_{\mathrm{EM}} = \varepsilon_0\varepsilon_r\mathbf{E} \otimes \mathbf{E} - \frac{1}{2}\varepsilon_0\varepsilon_r(\mathbf{E} \cdot \mathbf{E})I \qquad (5\text{-}3)$$

where the relative permittivity of the material with the absence of deformation is introduced as $\varepsilon_r = 1 + \chi_0$. It can be written equivalently as

$$\sigma_{\mathrm{EM}} = \sigma_{\mathrm{EM, V}} + \varepsilon_0\chi_0\left[\mathbf{E} \otimes \mathbf{E} - \frac{1}{2}(\mathbf{E} \cdot \mathbf{E})I\right]$$

where the two terms represent, respectively, the contributions from: the underlying free space and the material polarization.

📋 | The Electromechanical Forces coupling feature applies the material and underlying free space contributions to the stress as a body load. In weak form: $-\sigma_{\mathrm{EM}}:\mathrm{test}(\mathrm{T})$.

LARGE DEFORMATIONS

For finite deformations, the expressions for the electromagnetic stress and material polarization can be derived using the following thermodynamic potential called electric enthalpy:

$$H_{\mathrm{eme}} = W_s(C) - \frac{1}{2}[\varepsilon_0\varepsilon_r JC^{-1}]:(\mathbf{E}_{\mathrm{m}} \otimes \mathbf{E}_{\mathrm{m}})$$

where the subscript m indicates that the vector components must be taken on the material frame, and the right Cauchy-Green deformation tensor is

$$C = F^T F$$

with $F = \nabla \mathbf{u} + I$, and $J = \det(F)$. The mechanical energy function $W_s(C)$ depends on the solid model used.

The total second Piola-Kirchhoff stress tensor is given by

$$S = 2\frac{\partial H_{eme}}{\partial C}$$

and the electric displacement can be calculated as

$$\mathbf{D}_m = -\frac{\partial H_{eme}}{\partial \mathbf{E}_m}$$

The electromagnetic stress tensor can be used to compute the forces acting on a dielectric body. From Equation 5-1 the balance of forces at the surface of a dielectric body (material 1) in vacuum or air (material 2) implies:

$$(\sigma_2 - \sigma_1)\mathbf{n} = \mathbf{0}$$

where the total stress tensors in media 1 and 2 are, respectively:

$$\sigma_1 = \sigma_{EM} + \sigma_{mech}$$
$$\sigma_2 = \sigma_{EM,V} - p_2 I$$

where σ_m is a mechanical component of the total stress and the ambient pressure p_2 in to surrounding air or other medium (if present). Using Equation 5-2:

$$\sigma_1\mathbf{n} = -p_2\mathbf{n} - \left(\frac{1}{2}\varepsilon_0\mathbf{E}\cdot\mathbf{E}\right)\mathbf{n} + \varepsilon_0(\mathbf{n}\cdot\mathbf{E})\mathbf{E}$$

where the electric field $\mathbf{E}$ is computed in material 2.

The Electromechanical Forces coupling feature applies the above defined traction as a boundary load on the boundaries which are external for the coupling feature selection but internal for the corresponding Electrostatics interface.

COMSOL Multiphysics does not explicitly include the ambient pressure definition on the Electromechanical Forces coupling feature. However, it is possible to add an additional surface force to the corresponding Solid Mechanics interface if the pressure is known or computed by another physics interface.

ABOUT DIFFERENT STRESS FORMS

There exists a long-lasting controversy in scientific literature about the definition of electromagnetic forces acting on solids. An extensive review can be found in Ref. 3. Many classic textbooks (for example, Ref. 1 and Ref. 2) operate with the so-called Minkowski stress tensor that is usually written as:

$$\sigma_{EM} = \mathbf{D} \otimes \mathbf{E} - \frac{1}{2}(\mathbf{D} \cdot \mathbf{E})I$$

The corresponding electromagnetic body force can be written as

$$\mathbf{f} = \nabla \cdot \sigma_{EM} = \rho_e \mathbf{E} - \frac{1}{2}(\mathbf{E} \otimes \mathbf{E}):\nabla\chi$$

which sometimes is referred to as the Korteweg-Helmholtz force. For homogeneous materials without deformation, one has $\nabla\chi = 0$. Hence, in the absence of free charges ($\rho_e = 0$), the body force becomes zero. Thus, the whole electromechanical load on the solid is due to the Maxwell stress jumps at the boundaries between domains with different material properties.

There exists two most often used alternatives. The first one is called Einstein-Laub stress tensor:

$$\sigma_{EM} = \mathbf{D} \otimes \mathbf{E} - \frac{1}{2}(\varepsilon_0 \mathbf{E} \cdot \mathbf{E})I$$

This form is widely accepted in modern electroelasticity and material science, see Ref. 4. The corresponding body force can be written as:

$$\mathbf{f} = \nabla \cdot \sigma_{EM} = \rho_e \mathbf{E} + (\mathbf{P} \cdot \nabla)\mathbf{E}$$

which is also called Kelvin force. Note that it is nonzero as soon as there are electric field variation and polarization within the material:

$$\mathbf{P} = \varepsilon_0 \chi \mathbf{E}$$

The other alternative is Chu stress tensor:

$$\sigma_{EM} = \varepsilon_0 \mathbf{E} \otimes \mathbf{E} - \frac{1}{2}(\varepsilon_0 \mathbf{E} \cdot \mathbf{E})I \tag{5-4}$$

and the corresponding electromagnetic body force is given by

$$\mathbf{f} = \nabla \cdot \sigma_{EM} = \rho_e \mathbf{E} - (\nabla \cdot \mathbf{P})\mathbf{E}$$

Note that in contrast to the above equation, the expression for the stress tensor itself (Equation 5-4) seems to ignore the material polarization properties. However, if the frame difference is important because of the deformations, one has

$$\mathbf{D}_m = \varepsilon_0 J C^{-1} \mathbf{E}_m + \varepsilon_0 \chi_m \mathbf{E}_m \tag{5-5}$$

The electric enthalpy can be written as

$$H_{eme} = W_s(C) - \frac{1}{2}\mathbf{D}_m \cdot \mathbf{E}_m$$

Then,

$$\mathbf{D}_m = -\frac{\partial H_{eme}}{\partial \mathbf{E}_m}$$

and the electromechanical second Piola-Kirchhoff stress tensor is given by

$$S_{EM} = 2\frac{\partial H_{eme}}{\partial C} = JF^{-1}\sigma_{EM}F^{-T} \tag{5-6}$$

where σ_{EM} is given by Equation 5-4, which shows the consistency of the theory. The key assumption behind the derivation of Equation 5-6 is that the material components of the electric susceptibility χ_m are independent of deformations.

📝 COMSOL Multiphysics provides a choice of the electromagnetic stress model, which is available in the Electromechanical Forces coupling feature. Any of the above presented three options can be selected; the default choice is the Minkowski stress tensor.

References for the Electromechanics Interface

1. J.A. Stratton, *Electromagnetic Theory*, Cambridge, MA, 1941.

2. L.D. Landau and E.M. Lifshitz, *Electrodynamics of Continuous Media*, Pergamon Press, pp. 69–73, 1960.

3. D.J. Griffiths, "Resource Letter EM-1: Electromagnetic Momentum," *Am. J. Phys.*, vol. 80, pp. 7–18, 2012.

4. L. Dorfmann, R.W. Ogden, *Mechanics and Electrodynamics of Electroelastic and Magnetoelastic Interactions*, Springer, 2014.

The Electromechanics, Boundary Elements Interface

The **Electromechanics, Boundary Elements** interface (⊞), found under the **Electromagnetics-Structure Interaction** ( ) branch within the **Structural Mechanics** branch ( ) or the **Electromagnetics and Mechanics** ( ) branch within the **AC/DC** branch ( ) when adding a physics interface, combines Solid Mechanics and Electrostatics, Boundary Elements to model the deformation of electrostatically actuated structures. The **Electromechanics, Boundary Elements** interface is available for 3D and planar 2D geometries.

When this multiphysics interface is added using the **Model Wizard**, a **Solid Mechanics** interface and an **Electrostatics, Boundary Elements** interface are added to the **Model Builder**. In addition, the **Multiphysics Couplings** node is added, which automatically includes the multiphysics coupling feature **Electromechanical Interface**.

Electromechanical Interface

Use **Electromechanical Interface** (⊞) boundary multiphysics coupling to model electromechanical interaction between deformable solid domains and nonsolid voids, where the electric field is computed using the boundary element method. The backward coupling to Electrostatics is due to the deformations of the boundaries.

🗒	The presence of such multiphysics node in the model will enforce the **Include geometric nonlinearity** option on all applicable study nodes.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is eme1.

Applicable boundaries are those that are both external boundaries to the domains selected under the **Solid Mechanics** interface and boundaries to the domains selected under the **Electrostatics, Boundary Elements** interface. In particular, all boundaries external to **Solid Mechanics** selection are applicable if the selection of the **Electrostatics, Boundary Elements** contains **Infinite void**.

> For nonsolid dielectric domains, remove them from the **Solid Mechanics** interface selection but keep them selected in the **Electrostatics, Boundary Elements** interface selection.

The coupling is a boundary load on a structure caused by the Maxwell Stress at the selected boundaries. The backward coupling to Electrostatics is due to the change of the spatial position of the boundaries caused by the deformation of the structure.

> Only electrically linear and isotropic materials can be modeled using **Electrostatics, Boundary Elements**. For more details, see The Electromechanics, Boundary Elements Interface chapter of the *AC/DC Module User's Guide*.

COUPLED INTERFACES

This section defines the physics interfaces involved in this Electromechanical Forces coupling. The **Solid mechanics** and **Electrostatics, Boundary Elements** lists include all applicable physics interfaces.

The default values depend on how the node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.
- If it is added automatically when the physics interface is chosen in the **Model Wizard** or **Add Physics** window, then the two participating physics interfaces are selected.

You can also select **None** from either list to uncouple the coupling from a physics interface. If the physics interface is removed from the **Model Builder**, for example **Solid**

Mechanics is deleted, then the **Solid mechanics** list defaults to **None** as there is nothing to couple to.

🗐 If a physics interface is deleted and then added to the model again, then in order to reestablish the coupling, you need to choose the physics interface again from the lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present physics interface. See Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

The Pyroelectricity Interface

The **Pyroelectricity** interface ( ) combines Electrostatics and Heat Transfer in Solids interfaces together with the constitutive relationships required to model pyroelectricity. Both the direct pyroelectric and inverse electrocaloric effects can be modeled. This interface requires the MEMS Module license.

The **Piezoelectricity** interface is available for 3D, planar 2D, and axisymmetric 2D geometries.

When this multiphysics interface is added using the **Model Wizard**, an **Electrostatics** interface and a **Heat Transfer in Solids** interface are added to the **Model Builder**.

In addition, the **Multiphysics Couplings** node is added, which automatically includes a multiphysics coupling node Pyroelectricity.

The full functionality of the Electrostatics interfaces is accessible at the domain, boundary line, or point level in the geometry. The participating **Electrostatics** interface by default adds the **Charge Conservation** node with its selection set to all domains, and the material type set to **Solid**. This node serves to represent solid dielectric material domains. Any number of **Charge Conservation** nodes can be also added manually to the interface and configured to represent either solid or nonsolid (for example, air or free space) materials.

The **Heat Transfer in Solids** will use its default domain feature, **Solid**, where you enter the thermal properties of the material. All heat transfer functionality for modeling is also accessible to include surrounding solid or nonsolid domains. To model heat transfer by convection in nonsolid domains, you can add a **Fluid** domain feature.

You enter the pyroelectric coupling data on the Pyroelectricity multiphysics coupling node.

	• The Electrostatics Interface
Q	• The Heat Transfer in Solids Interface

Pyroelectricity

Use **Pyroelectricity** multiphysics node ( ) to simulate pyroelectric and electrocaloric effects coupling the variations of temperature and electric polarization in solid dielectrics.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `pye1`.

COUPLED INTERFACES

This section defines the physics involved in the Pyroelectricity multiphysics coupling. The **Electrostatics** and **Heat Transfer** drop-down menu lists include all applicable physics interfaces.

The default values depend on how the Electrostriction node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (macOS and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.

- If it is added automatically when a Pyroelectricity multiphysics interface has been selected in the **Model Wizard** or **Add Physics** window, then the participating Electrostatics and Heat Transfer in Solids interfaces are selected.

You can also select **None** from either list to uncouple the Pyroelectricity node from a physics interface. If the physics interface is removed from the **Model Builder**, for

example **Electrostatics** is deleted, then the list defaults to **None** as there is nothing to couple to.

	If a physics interface is deleted and then added to the model again, then in order to reestablish the coupling, you need to choose the physics interface again from the lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present interface. See Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

DOMAIN SELECTION

The domain selection is set by default to all domains, so that all applicable domains are selected automatically. Such domains represent an intersection of the applicable domains under the corresponding **Electrostatics** and **Heat Transfer in Solids** interfaces selected in the coupling feature.

In **Electrostatics** interface, the following two domains are applicable:

- **Charge Conservation**, if its material type input is set to **Solid**. Use this domain feature for solid dielectric materials, for which a linear dependency can be assumed for the electric polarization with respect to the applied electric field.

- **Charge Conservation, Piezoelectric**. Use this domain feature for solid linear piezoelectric materials.

- **Charge Conservation, Ferroelectric**. Use this domain feature for solid ferroelectric or nonlinear piezoelectric materials.

In **Heat Transfer in Solids** interface, the applicable domains are:

- **Solid**. Use this domain feature for solid pyroelectric and dielectric materials.

To model solid non-pyroelectric domain, remove such domains from the coupling feature selection.

COUPLING TYPE

From the list, choose one of these coupling types:

- **Pyroelectric effect** to include only the change in the material polarization caused by temperature variations.

$$\mathbf{P}_e = (T - T_{ref})\mathbf{p}_{ET}$$

- **Electrocaloric effect**, also known as inverse pyroelectric effect, to include only the heat source due to the polarization variation in time.

$$Q_p = -T\frac{\partial}{\partial t}(\mathbf{p}_{ET} \cdot \mathbf{E})$$

- **Fully coupled** (default) to include both the direct and inverse pyroelectric effects.

COUPLING SETTINGS

You enter the **Total pyroelectric coefficient** $\mathbf{p}_{ET}$ which is a vector (SI unit $C/m^2/K$) specified in the selected coordinate system. The default is to take the values **From material**.

You also can enter the **Reference temperature** T_{ref} with the default value of 293.15 K.

The total pyroelectric coefficient $\mathbf{p}_{ET}$ is a coupling coefficient measure at constant stress (unclamped) conditions. When this coupling feature is used as part of The Piezoelectricity and Pyroelectricity Interface, the following relation between different pyroelectric coefficients holds:

$$\mathbf{p}_{ET} = \mathbf{p}_{ES} + e_{ES}\alpha$$

where $\mathbf{p}_{ES}$ is the primary pyroelectric coefficient measured at constant strain (clamped), and the second term is usually called the secondary pyroelectric coefficient with α being the thermal expansion vector and e_{ES} being the piezoelectric coupling matrix. The software does all needed conversions automatically.

REFERENCES FOR PYROELECTRICITY

1. Newnham R.E., *Properties of Materials. Anisotropy, Symmetry, Structure.* Oxford University Press, New York, 2005.

The Piezoelectricity and Pyroelectricity Interface

The **Piezoelectricity and Pyroelectricity** interface ( ) combines Solid Mechanics, Electrostatics, and Heat Transfer in Solids together with the constitutive relationships required to model piezoelectric applications, in which the temperature variation and electric charge are coupled. It requires the MEMS Module license.

The **Piezoelectricity and Pyroelectricity** interface is available for 3D, planar 2D, and axisymmetric 2D geometries.

When this multiphysics interface is added using the **Model Wizard**, a **Solid Mechanics** interface, an **Electrostatics** interface, and a **Heat Transfer in Solids** interface are added to the **Model Builder**.

In addition, the **Multiphysics Couplings** node is added, which automatically includes three multiphysics coupling nodes: Piezoelectric Effect, Pyroelectricity, and Thermal Expansion.

The participating **Solid Mechanics** interface includes the default **Piezoelectric Material** feature with its selection set to all domains. The **Electrostatics** interface has a default **Charge Conservation, Piezoelectric** feature with similar settings. Such features can be also added manually to their corresponding interfaces similar to any other material model therein. The Piezoelectric Effect multiphysics coupling node can be active only on the selection, where both features Piezoelectric Material and Charge Conservation, Piezoelectric are active. You specify the piezoelectric coupling data together with both the structural and electrical material data under the Piezoelectric Material node.

All solid mechanics and electrostatics functionality for modeling is also accessible to include surrounding elastic solids or air domains. For example, add any solid mechanics material for other solid domain, a dielectric model for air (via Charge Conservation feature), or a combination. Note that in order to model a nonsolid dielectric domain, you need to remove such domain from the domain selection for the entire Solid Mechanics interface. This is because all material models under that interface represent solid materials (with the Linear Elastic Material node being always present and active in all those domains, where it is not explicitly overridden by any other material model).

The **Heat Transfer in Solids** will use its default domain feature, **Solid**, where you enter the thermal properties of the material. All heat transfer functionality for modeling is also accessible to include surrounding elastic solids or air domains. To model heat transfer by convection in nonsolid domains, you can add a **Fluid** feature. Remove all domains selected under such feature from the entire Solid Mechanics interfaces selection.

You enter the pyroelectric coupling data on the Pyroelectricity multiphysics coupling node, and the thermal expansion data on the Thermal Expansion multiphysics coupling node.

🔍	• The Solid Mechanics Interface
	• The Electrostatics Interface
	• The Heat Transfer in Solids Interface

The Thermoelasticity Interface

The **Thermoelasticity** multiphysics interface ( ) can be found under the
Thermal-Structure Interaction ( ) branch within the **Structural Mechanics**
branch ( ) when adding a physics interface using the Model Wizard or the Add
Physics window. The thermoelastic effect results in local cooling of material under
tension and heating of material that is compressed. Irreversible heat transfer occurs
between the warm and cool regions of the solid, producing mechanical losses, which
can be important, particularly for small structures.

When this multiphysics interface is added, the Model Builder tree structure is
populated with two physics interfaces (**Solid Mechanics** and **Heat Transfer in Solids**), and
the **Thermal Expansion** multiphysics coupling.

In the Settings window for the Solid Mechanics interface, the **Structural Transient
Behavior** is set to **Include inertial terms** by default. In the Settings window for Thermal
Expansion, under the section **Heat Sources**, the check box for **Thermoelastic damping** is
enabled by default.

On the Study Types for Thermoelasticity

Since this multiphysics interface is aimed at computing the thermoelastic damping of
MEMS structures under vibration, it only makes sense to analyze the model in the time
or frequency domain. Furthermore, for the frequency-domain study types, it is
necessary to first include a stationary study step in order to establish the baseline
temperature (the linearization point for the temperature field). Therefore, only the
following three study types are recommended for the Thermoelasticity multiphysics
interface: **Time Dependent**; **Thermal Perturbation, Eigenfrequency**; and **Thermal
Perturbation, Frequency Domain**.

On the Temperature Variable for Thermoelasticity

As mentioned above, for the frequency domain study types, there are two components
of the temperature variable: the baseline temperature and the harmonic temperature
(which varies around the baseline). To specify the baseline temperature at a boundary,

use the **Temperature** boundary condition. To specify the harmonic temperature at the boundary, right-click the **Temperature** node and select the **Harmonic Perturbation** subnode.

> ⊞ *Thermoelastic Damping in a MEMS Resonator*: Application Library path **MEMS_Module/Actuators/thermoelastic_damping_3d**

On the Constituent Physics Interfaces
The Solid Mechanics interface is intended for general structural analysis of 3D, 2D, or axisymmetric bodies. In 2D, plane stress or plane strain assumptions can be used. The Solid Mechanics interface is based on solving Navier's equations, and results such as displacements, stresses, and strains are computed.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation. A Solid model is active by default on all domains. All functionality for including other domain types, such as a fluid domain, is also available. The temperature equation defined in solid domains corresponds to the differential form of the Fourier's law that may contain additional contributions like heat sources.

On the Couplings
The coupling occurs on the domain level, where two physical effects occur:

• The temperature from the Heat Transfer interface acts as a thermal load for the Solid Mechanics interface, causing thermal expansion.

• The tension and compression from the Solid Mechanics interface acts as a heat source for the Heat Transfer interface, causing local cooling and heating.

These effects are implemented by the combination of the multiphysics coupling mentioned above, and the use of the temperature from the default model inputs.

COUPLING FEATURES
When physics interfaces are added from the Model Wizard or using the Add Physics window then the appropriate coupling feature is automatically added to the physics interface. However, the physics interfaces can be added one at a time, followed by the coupling features.

For example, if single **Solid Mechanics** and **Heat Transfer in Solids** interfaces are added, then COMSOL Multiphysics adds an empty **Multiphysics** node. You can then choose the **Thermal Expansion** coupling feature.

💡	Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics Couplings** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

💡	Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature
The Fluid-Solid Interaction Interface node is described in the section The Thermal Stress, Solid Interface in the *Structural Mechanics User's Guide*.

▦	*Thermoelastic Damping in a MEMS Resonator*: Application Library path **MEMS_Module/Actuators/thermoelastic_damping_3d**

Theory for the Thermoelasticity Interface

The Thermoelasticity Interface theory is described in this section:

- Background Thermoelasticity Theory
- References for the Thermoelasticity Interface

Background Thermoelasticity Theory

When an elastic rod is stretched reversibly and adiabatically its temperature drops. The drop in temperature compensates for the increase in entropy caused by the stress in the rod (since the process is reversible the entropy remains constant). The equations of thermoelasticity incorporate this phenomenon, together with the irreversible processes that occur in a vibrating rod (which result in net entropy increases). When a structure vibrates in a mode with both local compression and expansion there are always some losses associated with the irreversible heat conduction between the expanding areas that cool and the contracting areas that heat. These losses result in thermoelastic damping, which can be important, particularly in small vibrating structures, such as resonant MEMS devices.

> References 1 to 5 in the section References for the Thermoelasticity Interface provide useful background information.

The equations of thermoelasticity are derived from the first law of thermodynamics, which can be stated as follows:

$$dU = dQ' + dW' \qquad (5\text{-}7)$$

where dU is the change in internal energy, dQ' is the heat flow into the system (the prime indicates an inexact differential in this case) and dW' is the work done on the system. For a small part of a solid (sufficiently small that the stresses and strains are uniform), with an initial reference density, ρ_0, the first law can be rewritten in the following form (assuming that the differential changes occur between equilibrium states):

$$du = T_a ds + \frac{1}{\rho_0}\sigma{:}d\varepsilon \qquad\qquad (5\text{-}8)$$

where T_a is the absolute temperature, s is the entropy per unit mass, σ is the elastic part of the second Piola-Kirchhoff stress (in general a rank 2 tensor), ε is the material strain (also a tensor). In general the second Piola-Kirchhoff stress tensor, $\mathbf{p}$, must be split into elastic (σ) and inelastic (τ) parts such that:

$$\mathbf{p} = \sigma + \tau$$

The elastic part of the stress tensor, σ, does work $\sigma{:}d\varepsilon$ during a change in the strain. The inelastic part of the stress tensor, τ, generates heat at a rate $\tau{:}(d\varepsilon/d\tau)$ when the strain is changing and is identified with internal or material damping. These internal damping mechanisms are associated with microscopic phenomena such as dislocation movement.

From Equation 5-8 it is possible to make the following identifications for T_a and σ:

$$T_a = \left(\frac{\partial u}{\partial S}\right)_\varepsilon \qquad \sigma = \rho_0\left(\frac{\partial u}{\partial\varepsilon}\right)_S$$

Next the *entropy balance equation* must be derived. Since thermoelasticity involves irreversible processes, the assumption of equilibrium required to derive Equation 5-8 is no longer valid. Instead an assumption of "local" equilibrium is made. It is assumed that although the system is not in equilibrium, there exists within small elements a state of local equilibrium, for which the local entropy per unit mass, s, is the same function of the internal energy, strain, and particle number as it was in equilibrium. This assumption is commonly employed in the modeling of transport phenomena and is justified only by the validity of conclusions derived from it and by results obtained from specific microscopic models, for near-equilibrium situations. For a small volume element in the material frame Equation 5-8 can then be written as:

$$\rho_0 ds = \frac{1}{T_a}\rho_0 du - \frac{1}{T_a}\sigma{:}d\varepsilon$$

The rate of change of entropy can then be written as:

$$\rho_0\frac{ds}{dt} = \rho_0\frac{1}{T_a}\frac{du}{dt} - \frac{1}{T_a}\sigma{:}\frac{d\varepsilon}{dt} \qquad\qquad (5\text{-}9)$$

From the first law (Equation 5-7) the rate of change of internal energy is given by:

$$\rho_0 \frac{du}{dt} = \frac{dq}{dt} + \frac{dw}{dt}$$

where w is the work done per unit volume and q is the heat accumulated per unit volume. The heat accumulated can be written as the sum of the heat sources and the divergence in the material frame heat flux:

$$\frac{dq}{dt} = -(\nabla \cdot \mathbf{q}) + Q + \tau : \frac{d\varepsilon}{dt}$$

where Q represents the heat source per unit volume and τ is the inelastic part of the stress tensor. The rate of doing work (per unit reference volume) by a linear elastic material is given by the elastic part of the second Piola-Kirchhoff stress contracted with the rate of material strain. Per unit volume the following equation is obtained:

$$\frac{dw}{dt} = \sigma : \frac{d\varepsilon}{dt}$$

so Equation 5-9 reduces to:

$$\rho_0 \frac{ds}{dt} = -\frac{1}{T_a}\nabla \cdot \mathbf{q} + \frac{1}{T_a}Q + \frac{1}{T_a}\tau : \frac{d\varepsilon}{dt} \tag{5-10}$$

The definition of the material thermal conductivity gives:

$$\mathbf{q} = -\kappa \nabla T_a$$

where κ is the thermal conductivity, defined in the material frame.

Therefore the equation is:

$$T_a \rho_0 \frac{ds}{dt} = \nabla \cdot (\kappa \nabla T_a) + Q + \tau : \frac{d\varepsilon}{dt} \tag{5-11}$$

It is now necessary to derive an expression for the rate of change of entropy with respect to time. In order to do this an assumption of local equilibrium is used once again. Using Equation 5-8 the equation is written:

$$d\left(u - T_a s - \frac{1}{\rho_0}\sigma : \varepsilon\right) = -sdT_a - \frac{1}{\rho_0}\varepsilon : d\sigma$$

which defines a new *thermodynamic potential*, the Gibbs free energy per unit mass, given by:

$$g = u - T_a s + \frac{1}{\rho_0}\sigma{:}\varepsilon$$

Changes in the Gibbs free energy per unit mass take the form:

$$dg = -s\,dT_a - \frac{1}{\rho_0}\varepsilon{:}d\sigma$$

which leads to the relations:

$$s = -\left(\frac{\partial g}{\partial T_a}\right)_\sigma \qquad \varepsilon = -\rho_0\left(\frac{\partial g}{\partial \sigma}\right)_{T_a}$$

By differentiating each of the above equations a second time, it is possible to derive the following *Maxwell relation*:

$$\left(\frac{\partial s}{\partial \sigma}\right)_{T_a} = \frac{1}{\rho_0}\left(\frac{\partial \varepsilon}{\partial T_a}\right)_\sigma = -\frac{\partial^2 g}{\partial \sigma \partial T_a} \tag{5-12}$$

It is now possible to derive an expression for the entropy of the solid. Assuming that the elastic stress is an invertible function of the strain, we can write $s=s(\sigma,T_a)$. Thus:

$$ds = \left(\frac{\partial s}{\partial \sigma}\right)_{T_a}{:}d\sigma + \left(\frac{\partial s}{\partial T_a}\right)_\sigma dT_a$$

Using the Maxwell relation in [Equation 5-12](#):

$$ds = \frac{1}{\rho_0}\left(\frac{\partial \varepsilon}{\partial T_a}\right)_\sigma{:}d\sigma + \left(\frac{\partial s}{\partial T_a}\right)_\sigma dT_a$$

Therefore:

$$\frac{ds}{dt} = \frac{1}{\rho_0}\left(\frac{\partial \varepsilon}{\partial T_a}\right)_\sigma{:}\frac{d\sigma}{dt} + \left(\frac{\partial s}{\partial T_a}\right)_\sigma \frac{dT_a}{dt}$$

By definition the heat capacity of the solid at constant stress is given by:

$$c_p = \left(\frac{\partial q}{\partial T}\right)_\sigma = T_a\left(\frac{\partial s}{\partial T}\right)_\sigma$$

Thus:

$$\frac{ds}{dt} = \frac{1}{\rho_0}\left(\frac{\partial\varepsilon}{\partial T_a}\right)_\sigma:\frac{d\sigma}{dt} + \frac{c_p}{T_a}\frac{dT_a}{dt} \tag{5-13}$$

Substituting Equation 5-13 into Equation 5-11 gives the following equation for thermoelasticity:

$$\rho_0 c_p \frac{dT_a}{dt} = \nabla\cdot(\kappa\nabla T_a) + Q + \tau:\frac{d\varepsilon}{dt} - T_a\left(\frac{\partial\varepsilon}{\partial T_a}\right)_\sigma:\frac{d\sigma}{dt} \tag{5-14}$$

An additional heat source term is present in Equation 5-14, compared to the standard heat transfer equations in solids. This term couples the structural problem with the heat transfer problem. In turn the heat transfer equation couples back into the structural problem through the constitutive relationship.

In the particular case of a linear elastic material (in the absence of damping) the stress and strain are related by Duhamel-Hooke's law:

$$(\sigma - \sigma_i) = \mathbf{C}:(\varepsilon - \varepsilon_i - \alpha(T_a - T_{ref})) \tag{5-15}$$

where $\mathbf{C}$ is the elasticity tensor, σ_i is the initial stress, ε_i is the initial strain, and T_{ref} is the reference temperature at which the strain and stresses take the initial values.

This equation couples the heat transfer equation to the structural problem. Given a temperature independent thermal expansivity, and no material damping, Equation 5-14 takes the form:

$$\rho_0 c_p \frac{dT_a}{dt} = \nabla\cdot(\kappa\nabla T_a) + Q - T_a\alpha:\frac{d\sigma}{dt}$$

LINEARIZED EQUATIONS AND FREQUENCY DOMAIN FORMULATION

In the time domain it is convenient to linearize Equation 5-14, and this is necessary to solve problems in the frequency domain.

The independent variables in the system are T_a and $\mathbf{u}$ (where $\mathbf{u}$ is the displacement vector). Since the displacements drive the thermal equations it is natural to write the displacement vector as consisting of small deviations from its equilibrium value such that:

$$\mathbf{u} = \mathbf{u}_0 + a\mathbf{u}_1 + a^2\mathbf{u}_2 + \dots$$

where a is a constant that is subsequently set to 1 and that is used to keep track of the order of the term in the expansion.

The corresponding temperatures, stresses, and strains in the device are given by:

$$T_a = T_0 + aT_1 + a^2T_2 + \ldots$$

$$\varepsilon = \varepsilon_0 + a\varepsilon_1 + a^2\varepsilon_2 + \ldots$$

$$\sigma = \sigma_0 + a\sigma_1 + a^2\sigma_2 + \ldots$$

$$\tau = a_1\tau_1 + a^2\tau_2 + \ldots$$

where a is a constant that is subsequently set to 1. a can be used to track the order of the deviation from T_0.

Substituting these relations into Equation 5-14 and keeping terms up to the second order gives:

$$a\rho_0 c_p \frac{dT_1}{dt} + a^2 \rho_0 c_p \frac{dT_2}{dt} = \nabla \cdot (\kappa \nabla T_0) + a\nabla \cdot (\kappa \nabla T_1) + a^2 \nabla \cdot (\kappa \nabla T_2) + Q_0$$

$$+ a^2 \tau_1 : \frac{d\varepsilon_1}{dt} - aT_0 \left(\frac{\partial \varepsilon}{\partial T_a}\right)_\sigma : \frac{d\sigma_1}{dt}$$

$$- a^2 T_1 \left(\frac{\partial \varepsilon}{\partial T_a}\right)_\sigma : \frac{d\sigma_1}{dt} - a^2 T_0 \left(\frac{\partial \varepsilon}{\partial T_a}\right)_\sigma : \frac{d\sigma_2}{dt}$$

$$- a^2 T_0 T_1 \frac{\partial}{\partial T_a} \left(\frac{\partial \varepsilon}{\partial T_a}\right)_\sigma : \frac{d\sigma_1}{dt}$$

Note that the heat source term, Q, is assumed to be time independent ($Q=Q_0$). The final term in the above equation results from consideration of the Taylor expansion of the strain at constant stress, which to first order can be represented by a Taylor series of the form:

$$\varepsilon(T_a) = \varepsilon(T_0 + aT_1) = \varepsilon_0 + \left(\frac{\partial \varepsilon_0}{\partial T_a}\right) aT_1 + \ldots$$

Consequently:

$$\left(\frac{\partial \varepsilon}{\partial T_a}\right)_\sigma = \left(\frac{\partial \varepsilon_0}{\partial T_a}\right)_\sigma + aT_1 \frac{\partial}{\partial T_a} \left(\frac{\partial \varepsilon_0}{\partial T_a}\right)_\sigma + \ldots$$

Note that for the particular case of a linear elastic material:

$$\sigma_0 - \sigma_i = \mathbf{C} : (\varepsilon_0 - \varepsilon_i - \alpha(T_0 - T_{ref}))$$

$$a\sigma_1 + a^2\sigma_2 = \mathbf{C} : (a\varepsilon_1 + a^2\varepsilon_2 - a\alpha T_1 - a^2\alpha T_2)$$

$$a\tau_1 + a^2\tau_2 = i\eta\mathbf{C} : (a\varepsilon_1 + a^2\varepsilon_2 - a\alpha T_1 - a^2\alpha T_2)$$

where the material damping has been introduced into the equation system by means of an anisotropic loss factor, η.

Grouping together terms of the same order in a gives the following equations:

$$-\nabla \cdot (\kappa\nabla T_0) = Q_0$$

$$a\rho_0 c_p \frac{dT_1}{dt} = a\nabla \cdot (\kappa\nabla T_1) - aT_0 \left(\frac{\partial\varepsilon_0}{\partial T_a}\right)_\sigma : \frac{d\sigma_1}{dt}$$

$$a^2\rho_0 c_p \frac{dT_2}{dt} = a^2\nabla \cdot (\kappa\nabla T_2) + a^2\tau_1 : \frac{d\varepsilon_1}{dt} - a^2 T_1 \left(\frac{\partial\varepsilon_0}{\partial T_a}\right)_\sigma : \frac{d\sigma_1}{dt}$$

$$-a^2 T_0 T_1 \frac{\partial}{\partial T_a}\left(\frac{\partial\varepsilon_0}{\partial T_a}\right)_\sigma : \frac{d\sigma_1}{dt} - a^2 T_0 \left(\frac{\partial\varepsilon_0}{\partial T_a}\right)_\sigma : \frac{d\sigma_2}{dt}$$

The linear equation can be written in the form:

$$\rho_0 c_p \frac{dT}{dt} = \nabla \cdot (\kappa\nabla T) - T_0 \left(\frac{\partial\varepsilon_0}{\partial T_a}\right)_\sigma : \frac{d\sigma}{dt} \tag{5-16}$$

Where $a=1$ and the subscript has been dropped from the dependent variable T (previously T_1) and the stress deviation σ (previously σ_1). Equation 5-16 is solved by the Thermoelasticity interface. Writing the equation in the frequency domain gives:

$$i\omega\rho_0 c_p T = \nabla \cdot (\kappa\nabla T) - i\omega T_0 \left(\frac{\partial\varepsilon_0}{\partial T_a}\right)_\sigma : \sigma$$

This is the frequency domain form of the equations solved by COMSOL Multiphysics.

Considering the quadratic terms in Equation 5-12, there are four second-order heat source terms in the equation system:

$$Q_{visc} = \tau_1 : \frac{d\varepsilon_1}{dt}$$

$$Q_{TED1} = -T_1 \left(\frac{\partial \varepsilon_0}{\partial T} \right)_\sigma : \frac{d\sigma_1}{dt}$$

$$Q_{TED2} = -T_0 T_1 \frac{\partial}{\partial T} \left(\frac{\partial \varepsilon_0}{\partial T} \right)_\sigma : \frac{d\sigma_1}{dt}$$

$$Q_{TED3} = -T_0 \left(\frac{\partial \varepsilon_0}{\partial T} \right)_\sigma : \frac{d\sigma_2}{dt}$$

In the frequency domain, the first three terms involve the products of small harmonic deviations with other similar terms (that is, terms that vary in time in a similar manner to $\cos^2 \omega t$. The trigonometric identity $\cos^2 \omega t = (1 - \cos 2\omega t)/2$ makes it explicit that these terms have a time-independent heating effect in addition to the heating that occurs at double the frequency of the driving signal). These second-order terms have a small effect on the time-varying solution and lead to a second harmonic term in the frequency domain. However, if the time average is nonzero, this can, over many oscillatory periods, result in a change in the background temperature. In COMSOL Multiphysics the (nonzero) time-averaged heat sources are available as variables, so that the self heating of the structure can be computed.

First note that the time average of the term Q_{TED3} must be zero, since although the second order stress term σ_2 has a nonzero time average, its time derivative cannot. The other terms in the expression for Q_{TED3} are all time independent. Each of the remaining terms can have a nonzero time average as these involve the multiplication of two separate first-order terms (potentially with phase differences between them). The following result for the multiplication of two general oscillatory complex numbers, $a_1 e^{i(\omega t + \phi_1)}$ and, $a_2 e^{i(\omega t + \phi_2)}$ are useful:

$$\mathrm{Re}\left\{ a_1 e^{i\phi_1} e^{i\omega t} \right\} \mathrm{Re}\left\{ a_2 e^{i\phi_2} e^{i\omega t} \right\}$$

$$= a_1 a_2 \frac{1}{4} (e^{i(\omega t + \phi_1)} + e^{-i(\omega t + \phi_1)})(e^{i(\omega t + \phi_2)} + e^{-i(\omega t + \phi_2)})$$

$$= a_1 a_2 \frac{1}{4} (e^{i(\phi_1 - \phi_2)} + e^{-i(\phi_1 - \phi_2)} + e^{i(2\omega t + \phi_1 + \phi_2)} + e^{-i(2\omega t + \phi_1 + \phi_2)})$$

Taking the time average of the above expression gives:

$$\left\langle \mathrm{Re}\left\{a_1 e^{i\phi_1}e^{i\omega t}\right\}\mathrm{Re}\left\{a_2 e^{i\phi_2}e^{i\omega t}\right\}\right\rangle \; = \; a_1 a_2 \frac{1}{4}\left(e^{i(\phi_1-\phi_2)}+e^{-i(\phi_1-\phi_2)}\right)$$

$$\left\langle \mathrm{Re}\left\{a_1 e^{i\phi_1}e^{i\omega t}\right\}\mathrm{Re}\left\{a_2 e^{i\phi_2}e^{i\omega t}\right\}\right\rangle \; = \; \frac{1}{2}\mathrm{Re}\left\{a_1 e^{i\phi_1}e^{i\omega t}a_2 e^{-i\phi_2}e^{-i\omega t}\right\}$$

The time average heat sources above can be written in the following form in the frequency domain:

$$\langle Q_{visc}\rangle \; = \; \frac{1}{2}\mathrm{Re}\{\tau:(i\omega\varepsilon)^*\}$$

$$\langle Q_{TED1}\rangle \; = \; -\frac{1}{2}\mathrm{Re}\left\{T\left(\frac{\partial\varepsilon_0}{\partial T}\right)_\sigma:(i\omega\sigma)^*\right\}$$

$$\langle Q_{TED2}\rangle \; = \; -\frac{1}{2}\mathrm{Re}\left\{T_0 T\frac{\partial}{\partial T}\left(\frac{\partial\varepsilon_0}{\partial T}\right)_\sigma:(i\omega\sigma)^*\right\}$$

The derivation of the last term <Q_{TED2}> strongly relies on the prerequisite that the temperature dependence of the thermal expansion coefficient alpha is thermodynamically consistent. Since this prerequisite is usually not satisfied due to measurement or formulation errors, we have decided to remove the variable from the physics interface.

COMSOL Multiphysics also defines a total heat source term (`solid.Qh`) of the form:

$$\langle Q_h\rangle \; = \; \langle Q_{visc}\rangle + \langle Q_{TED1}\rangle$$

The `realdot` operator is used to evaluate these quantities in a manner that produces the correct terms in the Jacobian for sensitivity analyses. To compute the slow time dependence of the background temperature, the steady-state equation can be modified to allow for the time dependence of the background temperature and solved together with the frequency domain equations. In such a frequency-transient formulation, the following equations are solved:

$$\rho_0 c_p\frac{dT_0}{dt} \; = \; \nabla\cdot(\kappa\nabla T_0)+\langle Q_h\rangle + Q_0$$

$$i\omega\rho_0 c_p T = \nabla \cdot (\kappa \nabla T) - i\omega T_0 \left(\frac{\partial \varepsilon}{\partial T}\right)_\sigma : \sigma$$

With this formulation, the transient part of the problem can be solved on a time scale that is much longer than the period of the oscillations.

Operators, Functions, and Constants and Studies and Solvers in the *COMSOL Multiphysics Reference Manual*

References for the Thermoelasticity Interface

1. C.J. Adkins, *Equilibrium Thermodynamics*, Cambridge University Press, 1983.

2. W. Yourgrau, A. van der Merwe, and G. Raw, *Treatise on Irreversible and Statistical Thermodynamics: An Introduction to Nonclassical Thermodynamics*, Dover, 2002.

3. M. E. Gurtin, E. Fied, and L. Anand, *The Mechanics and Thermodynamics of Continua*, Cambridge University Press, 2010.

4. V.A. Lubarda, "On Thermodynamic Potentials in Linear Thermoelasticity", *International Journal of Solids and Structures*, vol. 41, no. 26, pp. 7377–7398, 2004.

5. A. Duwel, R. N. Candler, T. W. Kenny, and M. Varghese, "Engineering MEMS Resonators with Low Thermoelastic Damping", *Journal of Microelectromechanical Systems*, vol. 15, no. 6, 2006.

6. B. A. Auld, *Acoustic Fields and Waves in Solids*, Krieger Publishing Company, 1990.

6

Piezoresistivity Interfaces

This chapter describes the physics interfaces found under the **Structural Mechanics>Piezoresistivity** subbranch ( ).

In this chapter:

- The Piezoresistivity, Domain Currents Interface
- The Piezoresistivity, Boundary Currents Interface
- The Piezoresistivity, Shell Interface
- Theory for the Piezoresistivity Interfaces

The Piezoresistivity, Domain Currents Interface

The **Piezoresistivity, Domain Currents** predefined multiphysics coupling interface ( ), found under the **Structural Mechanics>Piezoresistivity** branch ( ) when adding a physics interface, combines the **Solid Mechanics** and **Electric Currents** interfaces to simulate the piezoresistive effect. This multiphysics interface is appropriate for situations when the thicknesses of the conducting and piezoresistive layers are resolved by the mesh.

When this multiphysics interface is added using the **Model Wizard**, a **Solid Mechanics** interface and an **Electric Currents** interface are added to the **Model Builder**.

In addition, the **Multiphysics Couplings** node is added, which automatically includes the multiphysics coupling feature **Piezoresistive Effect, Domain Currents**.

The participating **Electric Currents** interface by default adds the **Piezoresistive Material** node with its selection set to all domains. This node can also be added manually to the interface similar to any other material model therein. All piezoresistivity properties are entered in this node.

The **Piezoresistive Effect, Domain Currents** multiphysics coupling node can be active only on the selection where both features **Piezoresistive Material** and **Linear Elastic Material** (or **Hyperelastic Material**) are active.

| | In previous versions of COMSOL Multiphysics (up to version 5.2), a specific physics interface called **Piezoresistivity, Domain Currents** was added to the **Model Builder**. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see The Multiphysics Branch and Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*. |

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined couplings, for example **Piezoresistivity, Domain Currents**, specific settings are included with the physics interfaces and the coupling features.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if both **Solid Mechanics** and **Electric Currents** interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose the available coupling feature **Piezoresistive Effect, Domain Currents**, but the modified settings are not included. In this example, the **Piezoresistive Material** node needs to be added manually.

💡	Coupling features are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

💡	Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coordinate Systems

When setting up a model, the coordinate systems used for the participating physics interfaces should be consistent. In particular, the selected item for the **Coordinate system** menu under the **Solid Mechanics>Linear Elastic Material>Coordinate System Selection** section should match the one for the **Coordinate system** menu under the **Electric Currents>Piezoresistive Material>Coordinate System Selection** section.

Coupling Feature

The following multiphysics coupling feature node is described in this section:

- Piezoresistive Effect, Domain Currents

🔍	- Theory for the Piezoresistivity Interfaces

Piezoresistive Effect, Domain Currents

The **Piezoresistive Effect, Domain Currents** multiphysics coupling (📄) computes the appropriate conductivity tensor according to the mechanical stress or strain from the

Linear Elastic Material node in the **Solid Mechanics** interface and the piezoresistivity properties from the **Piezoresistive Material** node in the **Electric Currents** interface. The computed conductivity tensor is passed to the **Current Conservation** node in the **Electric Currents** interface.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is pzrd1.

DOMAIN SELECTION

Only domains that have both **Piezoresistive Material** selected in the **Electric Currents** interface and **Linear Elastic Material** selected in the **Solid Mechanics** interface are applicable.

COUPLED INTERFACES

This section defines the physics involved in the **Piezoresistive Effect, Domain Currents** multiphysics coupling. The **Solid mechanics** and **Electric Currents** lists include all applicable physics interfaces.

The default values depend on how the **Piezoresistive Effect, Domain Currents** node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.

- If it is added automatically when a **Piezoresistivity, Domain Currents** multiphysics coupling interface is selected in the **Model Wizard** or **Add Physics** window, then the participating **Solid Mechanics** and **Electric Currents** interfaces are selected.

You can also select **None** from either list to uncouple the **Piezoresistive Effect, Domain Currents** node from a physics interface. If the physics interface is removed from the

Model Builder, for example if **Solid Mechanics** is deleted, then the list defaults to **None** as there is nothing to couple to.

	If a physics interface is deleted and then added to the model again, then in order to reestablish the coupling, you need to choose the physics interface again from the lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present interface. See Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

The Piezoresistivity, Boundary Currents Interface

The **Piezoresistivity, Boundary Currents** predefined multiphysics coupling interface
( ), found under the **Structural Mechanics>Piezoresistivity** branch ( ) when adding
a physics interface, combines the **Solid Mechanics** and **Electric Currents, Shell** Interfaces
to simulate the piezoresistive effect. This multiphysics interface is appropriate for
situations in which the thickness of the piezoresistive layer is much less than that of the
structural layer.

When this multiphysics interface is added using the **Model Wizard**, a **Solid Mechanics**
interface and an **Electric Currents, Shell** interface are added to the **Model Builder**.

In addition, the **Multiphysics Couplings** node is added, which automatically includes the
multiphysics coupling feature **Piezoresistive Effect, Boundary Currents**.

The participating **Electric Currents, Shell** interface by default adds the **Piezoresistive
Material** node with its selection set to all domains. This node can also be added
manually to the interface similar to any other material model therein. All
piezoresistivity properties are entered in this node.

The **Piezoresistive Effect, Boundary Currents** multiphysics coupling node can be active
only on the selection where both features **Piezoresistive Material** and **Linear Elastic
Material** (or **Hyperelastic Material**) are active.

In previous versions of COMSOL Multiphysics (up to version 5.2), a
specific physics interface called **Piezoresistivity, Boundary Currents** was
added to the **Model Builder**. Now, a predefined multiphysics coupling
approach is used, improving the flexibility and design options for your
modeling. For specific details, see The Multiphysics Branch and
Multiphysics Modeling Workflow in the *COMSOL Multiphysics
Reference Manual*.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined couplings, for example
Piezoresistivity, Boundary Currents, specific settings are included with the physics
interfaces and the coupling features.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if both **Solid Mechanics** and **Electric Currents, Shell** interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose the available coupling feature **Piezoresistive Effect, Boundary Currents** but the modified settings are not included. In this example the **Piezoresistive Material** node needs to be added manually.

💡	Coupling features are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

💡	Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coordinate Systems
When setting up a model, the coordinate systems used for the participating physics interfaces should be consistent. In particular, the selected item for the **Coordinate system** menu under the **Solid Mechanics>Linear Elastic Material>Coordinate System Selection** section should match the one for the **Coordinate system** menu under the **Electric Currents, Shell>Piezoresistive Material>Coordinate System Selection** section.

Coupling Feature
The following multiphysics coupling feature node is described in this section:

• Piezoresistive Effect, Boundary Currents

🔍	• Theory for the Piezoresistivity Interfaces

▥	*Piezoresistive Pressure Sensor* Application Library path: **MEMS_Module/Sensors/piezoresistive_pressure_sensor**

Piezoresistive Effect, Boundary Currents

The **Piezoresistive Effect, Boundary Currents** multiphysics coupling node (⬗) computes the appropriate conductivity tensor according to the mechanical stress or strain from the **Linear Elastic Material** node in the **Solid Mechanics** interface and the piezoresistivity properties from the **Piezoresistive Material** node in the **Electric Currents, Shell** interface. The computed conductivity tensor is passed to the **Current Conservation** node in the **Electric Currents, Shell** interface.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is pzrb1.

BOUNDARY SELECTION

Only boundaries that have both **Piezoresistive Material** selected in the **Electric Currents, Shell** interface and are adjacent to domains with **Linear Elastic Material** selected in the **Solid Mechanics** interface are applicable.

COUPLED INTERFACES

This section defines the physics involved in the **Piezoresistive Effect, Boundary Currents** multiphysics coupling. The **Solid mechanics** and **Electric currents, shell** lists include all applicable physics interfaces.

The default values depend on how the **Piezoresistive Effect, Boundary Currents** node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.

- If it is added automatically when a **Piezoresistivity, Boundary Currents** multiphysics coupling interface is selected in the **Model Wizard** or **Add Physics** window, then the participating **Solid Mechanics** and **Electric Currents, Shell** interfaces are selected.

You can also select **None** from either list to uncouple the **Piezoresistive Effect, Boundary Currents** node from a physics interface. If the physics interface is removed from the **Model Builder**, for example if **Solid Mechanics** is deleted, then the list defaults to **None** as there is nothing to couple to.

	If a physics interface is deleted and then added to the model again, then in order to reestablish the coupling, you need to choose the physics interface again from the lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present interface. See Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

The Piezoresistivity, Shell Interface

> **!** This physics interface requires a Structural Mechanics Module license. For theory and physics interface feature descriptions relating to the Shell interface, see the *Structural Mechanics Module User's Guide*.

The **Piezoresistivity, Shell** predefined multiphysics coupling interface ( ), found under the **Structural Mechanics>Piezoresistivity** branch ( ) when adding a physics interface, is used when the structural layer is thin enough to be treated by the **Shell** interface, but the conducting and piezoresistive layers are still much thinner than the structural layers.

When this multiphysics interface is added using the **Model Wizard**, a **Shell** interface and an **Electric Currents, Shell** interface are added to the **Model Builder**.

In addition, the **Multiphysics Couplings** node is added, which automatically includes the multiphysics coupling feature **Piezoresistive Effect, Shell**.

The participating **Electric Currents, Shell** interface by default adds the **Piezoresistive Material** node with its selection set to all domains. This node can also be added manually to the interface similar to any other material model therein. All piezoresistivity properties are entered in this node.

The **Piezoresistive Effect, Shell** multiphysics coupling node can be active only on the selection where both a **Piezoresistive Material** and a **Linear Elastic Material** feature are active.

> **📄** In previous versions of COMSOL Multiphysics (up to version 5.2), a specific physics interface called **Piezoresistivity, Shell** was added to the **Model Builder**. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see The Multiphysics Branch and Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined couplings (for example, **Piezoresistivity, Shell**) specific settings are included with the physics interfaces and the coupling features.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if both **Shell** and **Electric Currents, Shell** interfaces are added, COMSOL adds an empty **Multiphysics Couplings** node. You can choose the available coupling feature **Piezoresistive Effect, Shell** but the modified settings are not included. In this example the **Piezoresistive Material** node needs to be added manually.

Coupling features are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coordinate Systems
When setting up a model, the coordinate systems used for the participating physics interfaces should be consistent. In particular, the selected item for the **Coordinate system** menu under the **Shell>Linear Elastic Material>Shell Local System>Coordinate System Selection** section should match the one for the **Coordinate system** menu under the **Electric Currents, Shell>Piezoresistive Material>Coordinate System Selection** section.

Coupling Feature
The following multiphysics coupling feature node is described in this section:

• Piezoresistive Effect, Shell

• Theory for the Piezoresistivity Interfaces

Piezoresistive Pressure Sensor, Shell Application Library path:
MEMS_Module/Sensors/piezoresistive_pressure_sensor_shell

Piezoresistive Effect, Shell

The **Piezoresistive Effect, Shell** multiphysics coupling node ( ) computes the appropriate conductivity tensor according to the mechanical stress or strain from the **Linear Elastic Material** node in the **Shell** interface and the piezoresistivity properties from the **Piezoresistive Material** node in the **Electric Currents, Shell** interface. The computed conductivity tensor is passed to the **Current Conservation** node in the **Electric Currents, Shell** interface.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is pzrs1.

BOUNDARY SELECTION

Only boundaries that have both **Piezoresistive Material** selected in the **Electric Currents, Shell** interface and **Linear Elastic Material** selected in the **Shell** interface are applicable.

COUPLED INTERFACES

This section defines the physics involved in the **Piezoresistive Effect, Shell** multiphysics coupling. The **Shell** and **Electric currents, shell** lists include all applicable physics interfaces.

The default values depend on how the **Piezoresistive Effect, Shell** node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.

- If it is added automatically when a **Piezoresistivity, Shell** multiphysics coupling interface is selected in the **Model Wizard** or **Add Physics** window, then the participating **Shell** and **Electric Currents, Shell** interfaces are selected.

You can also select **None** from either list to uncouple the **Piezoresistive Effect, Shell** node from a physics interface. If the physics interface is removed from the **Model Builder**, for example if **Shell** is deleted, then the list defaults to **None** as there is nothing to couple to.

	If a physics interface is deleted and then added to the model again, then in order to reestablish the coupling, you need to choose the physics interface again from the lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present interface. See Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

Theory for the Piezoresistivity Interfaces

In this section:

- Piezoresistive Materials
- Electric Currents Formulation
- Tensor Versus Matrix Formulations
- Material Property Definitions
- Coordinate Systems and Material Orientation
- References for the Piezoresistivity Interfaces

🔍
- The Piezoresistivity, Domain Currents Interface
- The Piezoresistivity, Boundary Currents Interface
- The Piezoresistivity, Shell Interface

❗ The Piezoresistivity, Shell interface uses features from the Shell interface, which requires the Structural Mechanics Module. Both the theory and features are described in the *Structural Mechanics Module User's Guide*.

Piezoresistive Materials

A piezoresistive material responds to an applied stress with a change in its resistivity. Normally piezoresistive materials are semiconductors, and the effect is associated with band structure changes that alter the carrier mobility and number density much more significantly than an applied stress would change, for example, the resistance of a similar metal. The equivalent elastoresistance effect relates the strain in the material to a change in resistivity. Note that the dependence between the structural mechanics and the electrical properties of the material is unidirectional, so that applied currents do not induce changes in the material stress or strain (assuming that other effects, such as heating of the material, are negligible).

Within a piezoresistive material, the relation between the electric field, **E**, and the current, **J**, becomes:

$$\mathbf{E} = \rho \cdot \mathbf{J} + \Delta\rho \cdot \mathbf{J} \tag{6-1}$$

where ρ is the resistivity and $\Delta\rho$ is the induced change in the resistivity. In the general case both ρ and $\Delta\rho$ are rank 2 tensors (matrices). The change in resistance is related to the stress, σ (for the piezoresistance form of the equations), or the strain, ε (for the elastoresistance form of the equations), by the constitutive relationship:

$$
\begin{aligned}
\Delta\rho &= \Pi \cdot \sigma \qquad \text{piezoresistance form} \\
\Delta\rho &= M \cdot \varepsilon \qquad \text{elastoresistance form}
\end{aligned}
\tag{6-2}
$$

where Π is the piezoresistance tensor (SI unit: $Pa^{-1}\Omega m$) and M is the elastoresistance tensor (SI unit: Ωm). Both of these quantities are material properties. Π and M are in this case rank 4 tensors; however, they can be represented as matrices if the resistivity, stress, and strain are converted to vectors within a reduced subscript notation.

🔍   Tensor Versus Matrix Formulations

Electric Currents Formulation

The flow of electric current through a conducting medium in the quasistatic case is governed by the equation of continuity which can be written in the form:

$$\nabla \cdot \left(\mathbf{J} - \mathbf{J}_e + \frac{\partial}{\partial t}\mathbf{D} \right) = Q_j$$

where $\mathbf{J}$ is the induced electric current, $\mathbf{J}_e$ includes external current sources that do not result from the material conductivity (such as thermoelectric currents), $\mathbf{D}$ is the electric displacement field, and Q_j is a current source term. Both Q_j and $\mathbf{J}_e$ are usually zero for piezoelectric devices.

In the stationary or quasistatic case, the electric current is related to the electric field, $\mathbf{E}$, by the constitutive relation:

$$\mathbf{J} = \sigma_{(c)} \cdot \mathbf{E} \tag{6-3}$$

where $\sigma_{(c)}$ is the conductivity tensor (the subscript c is used here to distinguish the conductivity from the strain).

Relating $\mathbf{E}$ and $\mathbf{D}$ to the electric potential ($\mathbf{E}=-\nabla V$, $\mathbf{D}=\varepsilon_0\varepsilon_{(r)}\mathbf{E}$, where ε_0 is the permittivity of free space and $\varepsilon_{(r)}$ is the relative permittivity tensor) gives:

$$-\nabla \cdot \left(\sigma_{(c)} \cdot \nabla V - \mathbf{J}_e + \frac{\partial}{\partial t}(\varepsilon_0 \varepsilon_{(r)} \cdot \nabla V) \right) = Q_j \qquad (6\text{-}4)$$

If current flow is restricted to a thin layer on the surface of a material this equation takes the form:

$$-\nabla_t \cdot \left(d_s \left(\sigma_{(c)} \cdot \nabla_t V - \mathbf{J}_e + \frac{\partial}{\partial t}(\varepsilon_0 \varepsilon_{(r)} \cdot \nabla_t V) \right) \right) = Q_j \qquad (6\text{-}5)$$

where the operator ∇_t represents the tangential derivative along the thin layer surface. Equation 6-4 and Equation 6-5 form the basis of the Electric Currents interface.

The constitutive relationship described by Equation 6-3 uses the conductivity form rather than the resistivity form used in Equation 6-1.

Since the equations of piezoelectricity are by convention formulated in the resistivity form, Equation 6-1 must be inverted to give:

$$\mathbf{J} = (\sigma_{(c)}^{-1} + \Delta \rho)^{-1} \cdot \mathbf{E} \qquad (6\text{-}6)$$

Although this expression appears cumbersome, it is required in this form for the general case where the conductivity and change in resistivity are nonsymmetric tensor quantities. For convenience, the tensor $\sigma_{(c,\text{ eff})}$ is defined in the interfaces (for example, for The Piezoresistivity, Domain Currents Interface as: `pzrd.sigmaceffXX`, `pzrd.sigmaceffXY`, and so forth) in the following manner:

$$\sigma_{(c,\text{eff})} = (\sigma_{(c)}^{-1} + \Delta \rho)^{-1} \qquad (6\text{-}7)$$

The Piezoresistivity, Domain Currents Interface solves a modified form of Equation 6-4 on the domain level:

$$-\nabla \cdot \left(\sigma_{(c,\text{eff})} \cdot \nabla(V) - \mathbf{J}_e + \frac{\partial}{\partial t}(\varepsilon_0 \varepsilon_{(r)} \cdot \nabla V) \right) = Q_j$$

while The Piezoresistivity, Boundary Currents Interface and The Piezoresistivity, Shell Interface solve a modified form of equation Equation 6-5 on the boundary level:

$$-\nabla_t \cdot \left(d_s \left(\sigma_{(c,\text{eff})} \cdot \nabla_t(V) - \mathbf{J}_e + \frac{\partial}{\partial t}(\varepsilon_0 \varepsilon_{(r)} \cdot \nabla_t V) \right) \right) = Q_j$$

where $\sigma_{(c, \text{eff})}$ is computed from Equation 6-7 and from the definition of $\Delta\rho$ given in Equation 6-2.

<table>
<tr><td>Q</td><td>• The Electric Currents Interface
• Theory for the Electric Currents Interface</td></tr>
</table>

Tensor Versus Matrix Formulations

The constitutive relations defined in Equation 6-2 relate the change in resistivity matrix (a rank 2 tensor) to the stress or strain matrix (also a rank 2 tensor). This means that the piezoresistivity tensor and the elastoresistance tensor are both rank 4 tensors. A formal tensor formulation of Equation 6-1 therefore takes the form:

$$(E_i = \rho_{ij}J^j + \Pi_{ijkl}J^j\sigma^{kl})\text{piezoresistance form}$$
$$E_i = \rho_{ij}J^j + M_{ij}^{kl}J^j\varepsilon_{kl} \qquad \text{elastoresistance form} \tag{6-8}$$

Here the Einstein summation convention is used. Note that this equation implies that $\Delta\rho_{ij}=\Pi_{ijkl}\sigma^{kl}$ and $\Delta\rho_{ij}=M_{ij}^{kl}\varepsilon_{kl}$. The tensors Π and M have a number of symmetry properties, that result from the fact that both σ, ε, and $\Delta\rho$ are symmetric and can be represented in the following form:

$$\sigma = \begin{bmatrix} \sigma^{xx} & \sigma^{xy} & \sigma^{xz} \\ \sigma^{xy} & \sigma^{yy} & \sigma^{yz} \\ \sigma^{xz} & \sigma^{yz} & \sigma^{zz} \end{bmatrix} \qquad \varepsilon = \begin{bmatrix} \varepsilon_{xx} & \varepsilon_{xy} & \varepsilon_{xz} \\ \varepsilon_{xy} & \varepsilon_{yy} & \varepsilon_{yz} \\ \varepsilon_{xz} & \varepsilon_{yz} & \varepsilon_{zz} \end{bmatrix} \qquad \Delta\rho = \begin{bmatrix} \Delta\rho_{xx} & \Delta\rho_{xy} & \Delta\rho_{xz} \\ \Delta\rho_{xy} & \Delta\rho_{yy} & \Delta\rho_{yz} \\ \Delta\rho_{xz} & \Delta\rho_{yz} & \Delta\rho_{zz} \end{bmatrix}$$

As a result of this symmetry, it is possible to represent σ, ε, and $\Delta\rho$ as six component vectors. There are two conventions for ordering the components in these vectors and the Piezoresistivity interfaces all use the Voigt notation prevalent in the literature.

<table>
<tr><td>📄</td><td>The Structural Mechanics branch interfaces use a distinct *standard notation* by default but does also support the Voigt notation.</td></tr>
</table>

Cubic materials, such as silicon, possess symmetries that mean the two notations are, in any case, equivalent when defining material properties. Within Voigt notation the vectors, s, e, and Δr, which correspond to σ, ε, and $\Delta\rho$ are:

$$s = \begin{bmatrix} \sigma^{xx} \\ \sigma^{yy} \\ \sigma^{zz} \\ \sigma^{yz} \\ \sigma^{xz} \\ \sigma^{xy} \end{bmatrix} = \begin{bmatrix} s_1 \\ s_2 \\ s_3 \\ s_4 \\ s_5 \\ s_6 \end{bmatrix} \qquad e = \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ 2\varepsilon_{yz} \\ 2\varepsilon_{xz} \\ 2\varepsilon_{xy} \end{bmatrix} = \begin{bmatrix} e_1 \\ e_2 \\ e_3 \\ e_4 \\ e_5 \\ e_6 \end{bmatrix} \qquad \Delta r = \begin{bmatrix} \Delta\rho_{xx} \\ \Delta\rho_{yy} \\ \Delta\rho_{zz} \\ \Delta\rho_{yz} \\ \Delta\rho_{xz} \\ \Delta\rho_{xy} \end{bmatrix} = \begin{bmatrix} \Delta r_1 \\ \Delta r_2 \\ \Delta r_3 \\ \Delta r_4 \\ \Delta r_5 \\ \Delta r_6 \end{bmatrix}$$

This notation is known as the reduced subscript notation, since the number of subscripts required to represent the stress, strain, or change in resistivity is reduced from two to one. The relationship between the stress or strain and the resistivity change becomes:

$$\Delta r = \pi \cdot s \qquad \text{piezoresistance form}$$
$$\Delta r = m \cdot e \qquad \text{elastoresistance form}$$

Where π and m are now matrices related to the components of the corresponding Π and M tensors in the following manner:

$$\pi = \begin{bmatrix} \Pi_{1111} & \Pi_{1122} & \Pi_{1133} & 2\Pi_{1123} & 2\Pi_{1113} & 2\Pi_{1112} \\ \Pi_{2211} & \Pi_{2222} & \Pi_{2233} & 2\Pi_{2223} & 2\Pi_{2213} & 2\Pi_{2212} \\ \Pi_{3311} & \Pi_{3322} & \Pi_{3333} & 2\Pi_{3323} & 2\Pi_{3313} & 2\Pi_{3312} \\ \Pi_{2311} & \Pi_{2322} & \Pi_{2333} & 2\Pi_{2323} & 2\Pi_{2312} & 2\Pi_{2312} \\ \Pi_{1311} & \Pi_{1322} & \Pi_{1333} & 2\Pi_{1323} & 2\Pi_{1313} & 2\Pi_{1312} \\ \Pi_{1211} & \Pi_{1222} & \Pi_{1233} & 2\Pi_{1223} & 2\Pi_{1213} & 2\Pi_{1212} \end{bmatrix}$$

$$m = \begin{bmatrix} M_{11}^{11} & M_{11}^{22} & M_{11}^{33} & M_{11}^{23} & M_{11}^{13} & M_{11}^{12} \\ M_{22}^{11} & M_{22}^{22} & M_{22}^{33} & M_{22}^{23} & M_{22}^{13} & M_{22}^{12} \\ M_{33}^{11} & M_{33}^{22} & M_{33}^{33} & M_{33}^{23} & M_{33}^{13} & M_{33}^{12} \\ M_{23}^{11} & M_{23}^{22} & M_{23}^{33} & M_{23}^{23} & M_{23}^{13} & M_{23}^{12} \\ M_{13}^{11} & M_{13}^{22} & M_{13}^{33} & M_{13}^{23} & M_{13}^{13} & M_{13}^{12} \\ M_{12}^{11} & M_{12}^{22} & M_{12}^{33} & M_{12}^{23} & M_{12}^{13} & M_{12}^{12} \end{bmatrix}$$

Components of the Π and M tensors that do not appear in these matrices are always equal to one of those that does appear as a result of symmetry constraints.

> For the most general triclinic crystal groups, the π and m matrices are not necessarily symmetric, as is the case for the elasticity matrix in structural mechanics.

This is because the rank four tensors Π and M lack one of the symmetry properties of the elasticity tensor ($\Pi_{ijkl} \neq \Pi_{klij}$ and $M_{ij}^{kl} \neq M_{kl}^{ij}$ see Ref. 1 and Ref. 2 for details). Finally the relationship between π and m is given by:

$$m = \pi \cdot D \tag{6-9}$$

where D is the elasticity matrix (s=D·e), which is defined in Voigt notation as:

$$D = \begin{bmatrix} C^{1111} & C^{1122} & C^{1133} & C^{1123} & C^{1113} & C^{1112} \\ & C^{2222} & C^{2233} & C^{2223} & C^{2213} & C^{2212} \\ & & C^{3333} & C^{3323} & C^{3313} & C^{3312} \\ & & & C^{2323} & C^{2313} & C^{2312} \\ & \text{symmetric} & & & C^{1313} & C^{1312} \\ & & & & & C^{1212} \end{bmatrix}$$

Material Property Definitions

The definitions of the piezoresistance and elastoresistance material properties implied by Equation 6-1, Equation 6-2, and Equation 6-8 include the resistivity of the material implicitly. This approach is common in the theoretical literature (see for example Ref. 1 and Ref. 2), since it is essential for a generalized description of the phenomena. However, many engineering texts (see for example Ref. 3, Ref. 4, and Ref. 5) describe the material properties in terms of a relative change in resistivity. In this approach the quantities π' and m' are defined such that:

$$\begin{aligned} \pi &= \rho_0 \pi' \\ m &= \rho_0 m' \end{aligned} \tag{6-10}$$

where ρ_0 is the scalar resistivity of the undeformed material. Here π' has the units Pa^{-1} and m' is dimensionless. For comparison π has the units ΩmPa^{-1} and m has units Ωm.

Equation 6-10 only applies when the resistivity is isotropic in the unstressed case. Cubic materials, such as silicon, have an isotropic resistivity when unstressed and so can employ this approach. It has the advantage that the piezoresistive material properties do not change with doping level for lightly doped silicon (dopant concentrations of less than approximately 10^{16} cm^{-3}). In COMSOL Multiphysics the built-in material properties have been implemented so that the dopant number density is added in the interface as a model input and the resistivity and conductivity are computed automatically within the material. These quantities are then included as part of the definitions of π and m. This approach provides the maximum level of flexibility (enabling noncubic materials and nonuniform dopant densities to be straightforwardly modeled), without sacrificing the convenience of the approach usually adopted for cubic materials. When entering custom material properties for materials not present in the library, it is important to remember that the resistivity should be included in the piezoresistance or elastoresistance material properties entered in the material library.

	The piezoresistive material properties for single crystal silicon included with the MEMS module are based on Ref. 6 and Ref. 7. Those provided for polysilicon are based on Ref. 3 and Ref. 7.

	About the Material Libraries for the MEMS Module

Coordinate Systems and Material Orientation

The material orientation is defined in COMSOL Multiphysics by means of a local coordinate system. The local coordinate system is the coordinate system that corresponds to the physical property axes of the material, that is, the orthogonal coordinate system in which the material properties are defined (see section 4.3 of Ref. 2). For materials with orthogonal crystallographic axes, the physical property axes correspond to the crystallographic axes. For other crystals a definition of the orientation of these axes should be provided with the material properties — often the standard described in Ref. 2 is used. In COMSOL Multiphysics, the equations of solid mechanics are usually solved in the local coordinate system, whilst the electric currents

equations are solved in the global system. Appropriate coordinate transforms of the relevant physical quantities have been implemented in the Piezoresistivity interfaces.

Reduced subscript matrices and vectors do not obey the usual tensor transformation rules for coordinate transforms. The COMSOL Multiphysics software uses the quadratic transformation laws to transform these quantities between coordinate systems where this is necessary. See Ref. 1 for further details.

The local coordinate system for a material is set within the Linear Elastic Material, Piezoresistive Material, or Conductive Material features (or their equivalents). These features default to using the global system, but alternative systems can be defined by adding one of the coordinate system nodes to the Definitions node level of the Model Builder. Often the Rotated System feature is used to specify the new coordinate system in terms of Euler angles.

Converting from Miller indices and wafer flat orientations to the appropriate Euler angles is not straightforward and a carefully drawn diagram is recommended.

Coordinate Systems in the *COMSOL Multiphysics Reference Manual*

References for the Piezoresistivity Interfaces

1. R.F. Tinder, *Tensor Properties of Solids*, Morgan and Claypool Publishers, 2008.

2. R.E. Newnham, *Properties of Materials*, Oxford University Press, 2005.

3. M. Bao, *Analysis and Design Principles of MEMS Devices*, Elsevier B. V., 2005.

4. S.D. Senturia, *Microsystem Design*, Springer, 2000.

5. A. A. Barlian; W.-T. Park; J.R. Mallon, Jr.; A.J. Rastegar; and B.L. Pruitt, "Review: Semiconductor Piezoresistance for Microsystems", *Proceedings of the IEEE*, vol. 97, no. 3, 2009.

6. C.S. Smith, "Piezoresistance Effect in Germanium and Silicon", *Physical Review*, vol. 94, no. 1, pp. 42–49, 1957.

7. C. Jacoboni, C. Canali, G. Ottaviani, and A. Alberigi Quaranta, "A Review of Some Charge Transport Properties of Silicon", *Solid-State Electronics*, vol. 20, pp. 77–89, 1977.

7

Glossary

This Glossary of Terms contains finite-element modeling terms specific to the MEMS Module and its applications. For mathematical terms as well as geometry and CAD terms specific to the COMSOL Multiphysics® software and its documentation, see the glossary in the *COMSOL Multiphysics Reference Manual*. To find references in the documentation set where you can find more information about a given term, see the index.

Glossary of Terms

accelerometer A sensor that measures acceleration or gravitational force. See also *sensor*.

actuator A device or mechanism that produces mechanical motion.

ALE See *arbitrary Lagrangian-Eulerian method*.

arbitrary Lagrangian-Eulerian (ALE) method A technique to formulate equations in a mixed kinematical description. An ALE referential coordinate system is typically a mix between the material (Lagrangian) and spatial (Eulerian) coordinate systems.

aspect ratio The ratio of the dimensions of a device in different directions, for example, the ratio of height to width. See also *lateral aspect ratio* and *vertical aspect ratio*.

biosensor A general term for sensor devices that either detect biological substances or use antibodies, enzymes, or other biological molecules in their operation. Biosensors are a subcategory of chemical sensors.

bonding A process by which one substrate is firmly attached to the surface of another.

buckling The sudden collapse or reduction in stiffness of a structure under a critical combination of applied loads.

cantilever beam A beam with one end fixed and one end free.

capacitive sensor A sensor that produces a signal due to a change in its capacitance.

Cauchy stress The most fundamental stress measure defined as force/deformed area in fixed directions not following the body.

compliance matrix The inverse of the elasticity matrix. See *elasticity matrix*.

comb drive A MEMS device consisting of interdigitated fingers similar in appearance to two interlocking combs.

constitutive equations The equations formulating the stress-strain relationship of a material.

creep Time-dependent material nonlinearity that usually occurs in metals at high temperatures in which the effect of the variation of stress and strain with time is of interest.

damping Dissipation of energy in a vibrating structure. A common assumption is viscous damping, where the damping is proportional to the velocity. See also *Rayleigh damping*.

elastic deformation A nonpermanent deformation that recovers its shape completely upon the release of an applied stress.

elasticity matrix The matrix D relating strain to stresses:

$$\sigma = D\varepsilon$$

Eulerian Model described and solved in a coordinate system that is fixed (spatial). See also *Lagrangian* and *arbitrary Lagrangian-Eulerian method*.

frequency-response analysis An analysis solving for the steady-state response from a harmonic excitation. Typically a frequency sweep is performed, solving for many excitation frequencies.

Green-Lagrange strain A measure of nonlinear strain used in large-deformation analysis. In a small-strain large rotation analysis, the *Green-Lagrange strain* corresponds to the engineering strain with the strain values interpreted in the original directions. The *Green-Lagrange strain* is a natural choice when formulating a problem in the undeformed state.

initial strain The strain in a stress-free structure before it is loaded. See also *strain* and *residual strain*.

initial stress The stress in a nondeformed structure before it is loaded. See also *stress* and *residual stress*.

Joule heating The increase in temperature of a medium as a result of resistance to an electric current flowing through it.

Knudsen number A dimensionless number that provides a measure of how rarefied a gas flow is, in other words, the average distance between the gas molecules compared to the length scale of the flow. The following equation defines the Knudsen number Kn where λ is the mean free path of the molecules and L is a length scale characteristic to the flow.

$$\text{Kn} = \frac{\lambda}{L}$$

Lagrangian Model described and solved in a coordinate system that moves with the material. See also *Eulerian* and *arbitrary Lagrangian-Eulerian method*.

large deformation The deformations are so large so the nonlinear effect of the change in geometry or stress stiffening need to be accounted for.

lateral aspect ratio The ratio of the length of a structure in the plane of a wafer to its width in that plane. See also *aspect ratio* and *vertical aspect ratio*.

MEMS An acronym for *microelectromechanical systems*. More generally, *MEMS* refers to systems, devices, and components with microscale size and in which the physics is not necessarily limited to electrical or mechanical phenomena.

microsystems See *MEMS*.

piezoelectricity The ability of certain crystalline materials to produce an electric voltage when subjected to mechanical stress (direct piezoelectric effect). Inversely, the material's ability to change shape when an external voltage is applied (reverse piezoelectric effect).

plane strain An assumption on the strain field where all out-of-plane strain components are assumed to be zero.

plane stress An assumption on the stress field where all out-of-plane stress components are assumed to be zero.

principle of virtual work States that the variation in internal strain energy is equal to the work done by external forces.

proof mass A predetermined test mass in a measurement device or machine that serves as the reference mass for the quantity to be measured.

Rayleigh damping A viscous damping model where the damping is proportional to the mass and stiffness through the mass and stiffness damping parameters.

residual strain Strain remaining in a structure after some operation, for example, the strain resulting from cooling a system after high-temperature bonding. It often appears as the initial strain for any consequent tasks. See also *strain* and *initial strain*.

residual stress Stress remaining in a structure after some operation, for example, stress resulting from cooling a system after high-temperature bonding. It often appears as the initial stress for any consequent tasks. See also *stress* and *initial stress.*

second Piola-Kirchhoff stress Conjugate stress to *Green-Lagrange strain* used in large deformation analysis.

sensor A device that measures a physical variable such as temperature or pressure and converts it (usually) to an electrical signal.

slide-film damping The damping effect of a fluid between two solid surfaces when the distance between them is small compared to their area and there is mostly tangential movement (lubrication). Slide-film damping is a type of *thin-film damping*.

spin tensor The skew-symmetric part of the velocity gradient tensor.

squeeze-film damping The damping effect of a fluid between two solid surfaces when the distance between them is small compared to their area and there is mostly normal movement. Squeeze-film damping is a type of *thin-film damping*.

strain Relative change in length, a fundamental concept in structural mechanics.

stress Internal forces in a material. Normal stresses are defined as forces/area normal to a plane, and shear stresses are defined as forces/area in the plane. A fundamental concept in structural mechanics.

thermoelastic damping When an elastic rod is stretched reversibly and adiabatically its temperature drops. The drop in temperature compensates for the increase in entropy caused by the stress in the rod (since the process is reversible the entropy remains constant). In a general deformation, the strain in a structure is nonuniform and so the temperature changes in a nonuniform manner throughout the structure. The irreversible flow of heat that accompanies these temperature changes result in energy loss and damping of the structure.

thin-film damping   Damping that occurs in a thin channel of fluid (fluid film) between moving structures. See *squeeze-film damping* and *slide-film damping*.

transducer   A device that converts one type of energy to another. It often refers to a device that responds to a physical parameter and converts it to an electrical signal.

vertical aspect ratio   The ratio of the height of a structure perpendicular to a wafer's surface to its depth in the wafer's plane.

Index