Material Library

User's Guide

5.4

COMSOL

Material Library User's Guide

Version: COMSOL 5.4

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, 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 at 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/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM021201

Contents

Chapter 1: Introduction

Chapter 2: Using the Material Library

1

Introduction

Welcome to the Material Library, an add-on product that provides predefined material data, primarily as piecewise polynomial functions of temperature. The Material Library contains more than 20,000 property functions; these functions specify various material properties of over 2500 materials.

The Material Library is ideal for multiphysics couplings such as electrical-thermal analysis and structural-thermal analysis because most of the properties are available as functions of temperature.

The Material Library Environment

When working with the Material Library, it is the same as working with any other material database. Below are descriptions about the predefined material databases, the Material Library folders, and the windows and pages you work in to add any material to your model.

See Materials in the *COMSOL Multiphysics Reference Manual* for an overview of working with material properties, material databases, and the Material Browser.

About the Material Library

The Material Library stores the material data in folders. A search engine on the **Material Browser** makes it easy to find materials to add to models — you can search by name, UNS number, or DIN number.

The following is some basic information about the available material properties contained in the Material Library.

- The Material Library incorporates mechanical, thermal, and electrical properties primarily for solid materials.
- The material properties are described as a function of some variable, typically temperature, and focus on elastic and thermal properties.
- Where applicable, data is given for a material's solid, liquid, and vapor phases. A material can also contain data for multiple orientations or variations.
- The properties are analytic functions over a given interval of the argument.
- Smoothing is used to interpolate the values of the properties between different intervals. You can choose the smoothing settings in order to obtain continuous first and second derivatives of the property functions.
- Materials can be copied to a *User-Defined Library* where you can add and edit properties. You can also plot and inspect the definition of a function.
- The material property data in the Material Library is based on the Material Property Database (MPDB) from JAHM Software, Inc.
- For all properties contained in the Material Library, you can view the literature reference, notes, and reference temperature (where applicable) by first selecting a

material property and then on the **Material Browser**, under **Properties**, click a specific property. Then information, when available, displays under **Property reference**.

- Working with Materials
- The Material Browser Window
- The Add Material Window

Where Do I Access the Documentation and Application Libraries?

A number of internet 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 about 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 app. 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 **Home** toolbar, **Windows** menu, click ( ) **Applications Libraries**.
- 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 are 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/community
Events	www.comsol.com/events
COMSOL Video Gallery	www.comsol.com/video
Support Knowledge Base	www.comsol.com/support/knowledgebase

2

Using the Material Library

This chapter describes the material properties in the Material Library and how to use them in your COMSOL Multiphysics® models. It also contains information about using functions to define material properties.

Working with Materials

The Material Browser Window

The **Material Browser** window ( ) contains a number of databases with a broad collection of elastic, solid mechanics, electromagnetic, fluid, chemical, thermal, piezoelectric, and piezoresistive properties of materials. The number of material databases depends on which COMSOL products your license includes. Use the Material Browser to find predefined materials and add them to the Model Builder, or create a custom material library.

Win

To open the **Material Browser** :

- On the **Materials** toolbar, click **Browse Materials**.
- Right-click the **Materials** node ( ), and then select **Browse Materials**.
- From the **Home** toolbar, select **Windows>Material Browser**.

Mac

Linux

To open the **Material Browser** :

- On the **Model Toolbar**, click **Browse Materials**.
- Right-click the **Materials** node ( ), and then select **Browse Materials**.
- Select **Windows>Material Browser**.

The **Material Browser** is similar to The Add Material Window but it includes detailed property information about each material. From this window you can also create a new material library and import a material library. See Adding Materials to a Component for information about adding materials to your model's components (geometries). Click **Done** ( ) to close the **Material Browser** and add the materials in the **Added to model** list to the model. Click **Cancel** ( ), press Escape, or click in the main toolbar to exit the **Material Browser** without adding any materials.

You can browse all of the available material databases or search for specific materials. There is also a **Recent Materials** folder where you find the most recently used materials. **Search** a specific material by name (or, for the Material Library product, by UNS number or DIN number, which are listed in the Material Browser when available).

When browsing the material databases, in particular the **Material Library**, some materials include additional information — UNS number, DIN number, and composition.

As in Figure 2-1, the following information is included in the window to the right of the material tree. Navigate in the material tree and click a material to display the information.

Material availability is based on the type of COMSOL Multiphysics license. For example, if you have the MEMS Module, you have the **Built-In**, **Liquids and Gases**, **MEMS**, and **Piezoelectric** material libraries.

PROPERTIES

While browsing the databases, predefined material properties for the selected material are listed in a table in the columns **Property**, **Expression**, **Unit**, and the **Property group** to which the material property belongs. If **Property group** is empty, the material property is a **Basic** property.

Under **Property reference**, for the materials in the Material Library product, reference information about a material's properties appears when you click a property above.

INPUTS

For some materials, predefined function inputs are listed in a table in the columns **Input**, **Variable**, and **Unit**. Inputs appear for material properties defined using functions that require the input. Typical inputs are temperature and pressure, for temperature- and pressure-dependent material properties, respectively.

CREATE A NEW MATERIAL LIBRARY OR IMPORT A MATERIAL LIBRARY

Click the **New Material Library** button ( ) to open the **New Material Library** dialog box. You can also right-click a material and select **Add to New Library** ( ) to create a new material library and add that material to the new library. Go to Creating a New Material Library and Adding and Editing Materials in the *COMSOL Multiphysics Reference Manual*.

Click the **Import Material Library** button ( ) to open the **Choose Material Library** dialog box. Go to Importing a Material Library in the *COMSOL Multiphysics Reference Manual*.

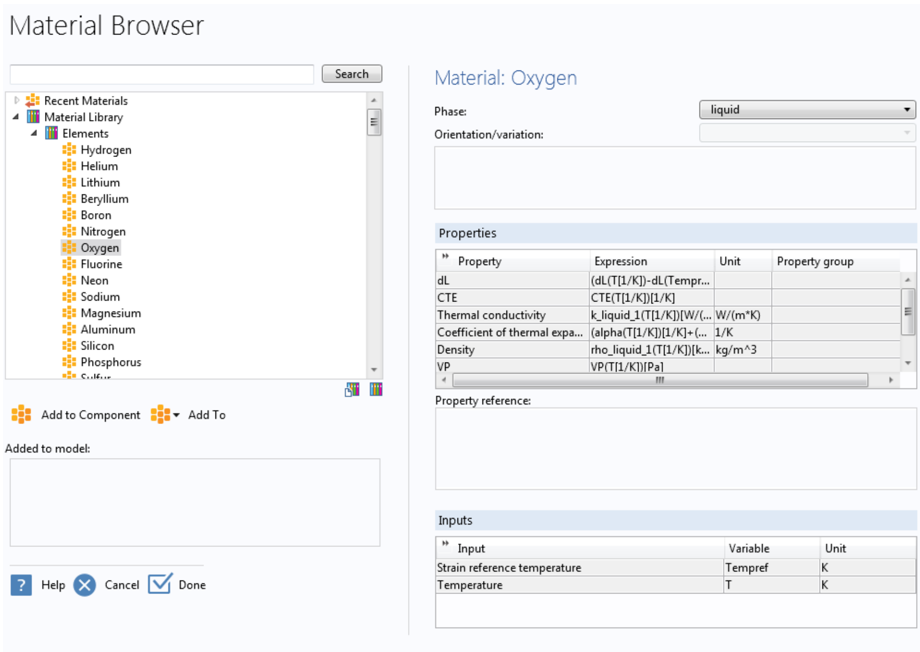


Figure 2-1: The Material Browser details a material's properties after selection. In this example, the properties of Oxygen are listed to the right of the Material Browser folders.

MATERIAL LIBRARY FOLDERS

TABLE 2-1: MATERIAL LIBRARY FOLDERS

FOLDER
Elements
Iron Alloys
Nickel Alloys
Aluminum Alloys
Copper Alloys
Magnesium Alloys
Titanium Alloys
Simple Oxides
Complex Oxides/Silicates
Carbides
Cermets
Tool Steels
Carbons
Thermal Insulators
Intermetallics and TBC
Refractory Metal Alloys
Nylons and Polyamides
Polyethers and Polyesters
Acetal Resins
PVDF (poly(vinylidene fluoride))
EVA (ethylene-vinyl acetate)
Miscellaneous Polymers
Polymer Composites
Elastomers
Epoxies
Minerals, Rocks, and Soils
Woods
Polypropylenes
PET Compounds
Controlled Expansion Alloys

TABLE 2-1: MATERIAL LIBRARY FOLDERS

FOLDER
Thermocouple Alloys
Semiconductors and Optical Materials
Organics and Hydrocarbons
Other Materials
Solders, Low Melting, and Dental
Cobalt Alloys
Resistance Alloys
Magnetic Alloys
Metal Matrix Composites
Ceramic Matrix Composites
Salts
Fuel Cell, Battery, and Electro-ceramics
Silicides
Borides
Glasses and Metallic Glasses
Nitrides and Beryllides
Cast Irons
Mold Materials

The Add Material Window

The **Add Material** window is similar to The Material Browser Window. It has the same material libraries available but does not include the detailed properties about each material. The number of material libraries depends on which COMSOL Multiphysics products your license includes. This window is a quick way to add materials to models.

To open the **Add Material** window :

- From the **Materials** toolbar, click **Add Material**.
- Right-click the **Materials** node ( ) and select **Add Material from Library**.

As in Figure 2-2you can browse all the available material databases or search for specific materials. There is also a **Recent Materials** folder where you find the most recently used materials. **Search** a specific material by name (or, for the Material Library product, by UNS number or DIN number).

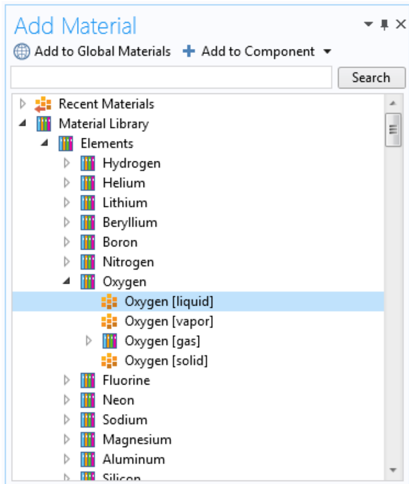


Figure 2-2: The Add Material window. In this example, the liquid phase of Oxygen is selected and can be added to the Material node in the local Component or as a global material in the Model Builder.

ADDING MATERIALS TO A COMPONENT

You can add materials to Component nodes using either the **Add Material** or **Material Browser** windows. In either window, use the **Search** field to find materials by name, UNS number, or DIN number. Or click any of the folders and subfolders to locate and add a specific material. To add a material to the current component, click the **Add to Component** button, right-click the material and choose **Add to Component**, or, in the **Add Material** window, press Enter. In the **Add Material** window you can also add a material to global **Materials** list and to the current selection. In the **Material Browser** window, you can also add the material to the global **Materials** list and to an existing user-defined or new material library.

For example, click the arrow to the left of **Elements** to expand that folder, and then click **Oxygen**.

In the **Add Material** window, all the materials are listed with a description of the phase and orientation/type next to the primary name (for example, **Oxygen [liquid]**, **Oxygen [vapor]**. This is different in the **Material Browser**, where you select these options from the **Phase** or **Orientation/variation** lists.

Using the Add Material Window

1 Open the **Add Material** window (see The Add Material Window).

2 In the **Add Material** window, select a material by phase (liquid, vapor, gas, or solid) and orientation/variation, when available.

3 Click the **Add to Global Materials** or **Add to Component** buttons, or right-click the material and select the same options from the context menu. If there is more than one **Component** node in the model tree, add the material to the applicable geometry.

- Click the **Add to Global Materials** button to add it under the global **Materials** node.
- Click the **Add to Component** button to add the material to the active component in the **Model Builder** and then make it an active material in the domains (or other geometric entities) where it is selected. You can also select any of the components in the model to add it to its **Materials** node, or select **Add to Switch 1**, for example, to add it under a **Switch** node for materials under the global **Materials** node. Right-click the **Material** node to rename it, for example, using the name of the material it represents.

Using the Material Browser Window

1 Open the **Material Browser** window (see The Material Browser Window).

2 In the **Material Browser**, select options from the **Phase** and **Orientation/variation** lists, when available (only included for some materials in the Material Library product). In this window you can review the material **Properties** and **Input** sections. See Viewing Material Property Information for information about viewing information about, for example, references for a specific material property.

3 Click the **Add to Component** button ( ) under the list of materials to add the selected material to the current model component. Alternatively, click the **Add To** button ( ) to add the material to the global **Materials** node (choose **Global Materials**), to any available model component, or to an existing or new user-defined material library. You can also right-click the selected material node to add that

material to a model component or user-defined material library. Materials that you have selected to add to any of the model components appear in the **Added to model** list.

4. Click **Done** ( ) to add the materials to the model tree in the **Model Builder** and close the **Material Browser**. If it is the first material in that model component, the material in the **Model Builder** becomes the default material; otherwise, the material is initially not used anywhere but becomes the active material in the domains (or other geometric entities) that you pick to add to that material's selection list.

Materials

Use the nodes under **Materials** ( ) to add predefined or user-defined materials, to specify material properties using model inputs, functions, values, and expressions as needed, or to create a custom material library. Also see Material Link, Switch for Materials, Working with External Materials, and About the Material Databases in the *COMSOL Multiphysics Reference Manual.*

You can right-click the **Materials** node and select **Add Materials from Library** to add a material using The Add Material Window or select Browse Materials to open The Material Browser Window for more thorough information about the available materials in the material libraries. Yous can also select **Blank Material** to add a Material node with no predefined material properties.

MATERIAL OVERVIEW

This section provides an overview of the materials in the Component node and where they are used. You can also add materials under **Global Definitions**. To access such global materials in a model component, use a Material Link.

The **Material** column lists the current materials in the Component using the materials' node labels from the model tree according to the settings defined in Displaying Node Names, Tags, and Types in the Model Builder.

The **Selection** column lists the geometric entities selected for the material (the domains, boundaries, or edges where the material is defined).

ERRORS RELATING TO THE MATERIAL NODES

If a material property in a physics interface takes its value from a material and no material is defined for the same geometric selection, a stop sign ( ) displays in the leftmost column and the **Material** column contains **Entities needing a material**. The **Selection** column contains the geometric entities in which a material definition is missing. The Materials node also indicates when there is a material error (see Figure 2-3). For example, if some property is deleted but needed in a part of the geometry, then the icon indicates where the error is located.

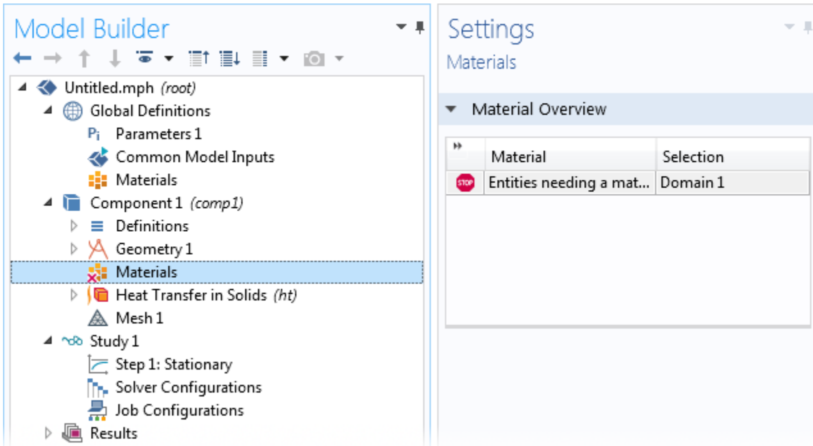


Figure 2-3: An example of a Materials node error.

The Settings Window for Material

The **Settings** window for **Material** ( ) summarizes the predefined or user-defined material properties for a material. This is where you can add or change material properties to fit your model and assign the material to all types of geometric entities: domains (most common), boundaries, edges (3D models only), or points. Also see Material Link and Switch for Materials.

After adding a material (see The Add Material Window and The Material Browser Window), click the Material node (for example, **Material 1** or **Copper**) in the **Model Builder**. The **Settings** window for **Material** opens.

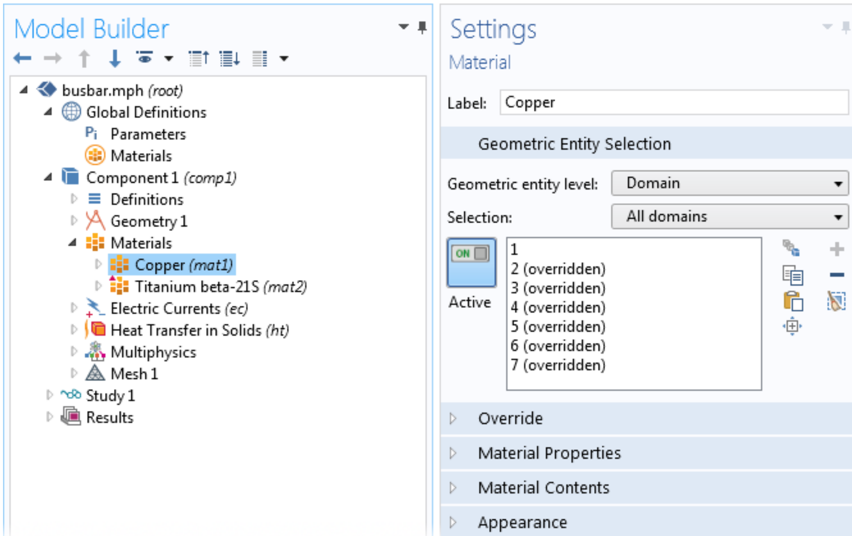


Figure 2-4: Click the Copper node to open the Settings window for Material for the node.

GEOMETRIC ENTITY SELECTION

Assign the material to some or all entities on a specific **Geometric entity level** — **Domain**, **Boundary**, **Edge** (3D only), or **Point** — on the geometry in the **Graphics** window (the geometry in the model).

By default, the first material in the Component is active in all domains (or all boundaries or edges if the Component only contains surfaces or edges). By assigning other materials to some or all domains, the first material is overridden and remains active only in domains where no other material, added below it in the **Materials** branch, is active.

If the Component contains features on different geometric entity levels, such as solid mechanics in domains coupled to beams on edges, and the features use the same material, you need to add two Material nodes with the same material, one defined in the domains, and the other defined on the edges.

OVERRIDE

This section shows if the material, in some or all parts of the geometry where it is active, is overridden by another material added underneath it in the Materials branch, or if it overrides another material above it.

The **Overridden by** list shows the names of the materials that override this material. The **Selection** list in the **Geometric Entity** section displays **(overridden)** for the geometric entities in which this material is overridden.

The **Overrides** list shows the names of the materials that this material overrides.

- Physics Exclusive and Contributing Node Types
- Physics and Variables Selection
- Physics Node Status

MATERIAL PROPERTIES

You can add material properties to the material if they are not already included. To do so, browse the available material property categories (**Basic Properties**, **Acoustics**, and so on), and select a material property or a collection of material properties in one of the property groups or material models that appear under the main level of material property categories. Right-click the material property or property group and select **Add to Material**, or click the **Add to Material** button ( ) to add the material property or group of properties to the material.

Review the properties listed in the **Material Contents** table before adding new material properties.

For example, under **Acoustics>Viscous Model** select **Bulk viscosity (muB)** and right-click to **Add to Material** or click the **Add to Material** button ( ). If you add a material model like the **Viscous Model** with more than one property, all of its material properties are added to the **Material Contents** table. In this example, a **Viscous model** node is added to the **Model Builder** and its associated properties are added to the **Material Contents** table.

To delete a property group, right-click the property group node (in the **Model Builder**) and select **Delete** ( ). The **Basic** property group cannot be deleted.

A Note About Adding Basic Material Properties

Material properties can be added to the **Basic** group or to any **User-Defined Property Group** from two locations — the **Settings** windows for **Material** and **Property Group**.

- When material properties are added from the **Basic** node's or a user-defined group node's **Settings** window for **Property Group**, they are listed under **Output Properties** and **Model Inputs** in that **Settings** window.
- When material properties are added from the **Settings** window for **Material**, the available material properties are listed under **Material Properties** and are added to the list under **Material Contents** with the property group listed. The list under **Material Contents** also contains material properties added from a subnode with a **Settings** window for **Property Group**.

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.

Simply put, **Solid** materials associate material properties with specific pieces of the material, and the properties follow the material as it moves around. In particular, a solid material may be inherently anisotropic, meaning that its axes rotate together with the material. The **Nonsolid** choice, in contrast, applies typically to liquids and gases whose properties are associated with fixed points in space and insensitive to local rotation of the material. Such materials are inherently isotropic when studied in isolation, but may exhibit anisotropy induced by external fields. In practice, this means that any anisotropic tensor properties in a Non-solid material must be functions of some external vector field.

MATERIAL CONTENTS

This section lists all of the material properties that are defined for the material or required by the physics in the model. The table lists the **Property**, **Variable**, **Value**, and **Unit** for the material property as well as the **Property group** to which the material property belongs. The **Property group** corresponds to the subnodes in the **Model Builder** with the same name. If required, edit the values or expression for the property's **Value**.

The left column provides visual cues about the status of each property:

- A stop sign ( ) indicates that an entry in the **Value** column is required. It means that the material property is required by a physics feature in the model but is undefined. When you enter a value in the **Value** column, the material property is added to its property group.
- A warning sign ( ) indicates that the material property has been added to the material but is still undefined. An entry is only required if the material property is to be used in the model.
- A green check mark ( ) indicates that the property has a **Value** and is currently being used in the physics of the model.
- Properties with no indication in the left column are defined but not currently used by any physics in the model.

You can change the value for any property by editing its value directly in the **Value** column, or, for a selected property, click the **Edit** button ( ) to enter a value in the window that opens. If the property can be anisotropic, you can choose to enter the values in one of these forms: **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic**. The **Variable** column lists the variable names corresponding to the degree of anisotropy. For example, for a symmetric electrical conductivity, it contains **{sigma11, sigma12, sigma22, sigma13, sigma23, sigma 33}; sigmaij = sigmaji**. For an isotropic electrical conductivity, it contains **sigma_iso; sigmaii = sigma_iso, sigmaij = 0**, where `sigma_iso` is the name of the variable for the isotropic electrical conductivity (available as, for example, `mat1.def.sigma_iso`).

APPEARANCE

The settings in this section make it possible to control or change the default appearance of a material in the Graphics window when working in the materials or physics parts of the model tree.

> In 3D components, the material is rendered including color and texture when **Scene Light** is active. In 2D models and in 3D components, when **Scene Light** is turned off, only a change of color is visible.

The **Family** list provides quick settings approximating the appearance of a number of common materials — **Air**, **Aluminum**, **Brick**, **Concrete**, **Copper**, **Gold**, **Iron**, **Lead**, **Magnesium**, **Plastic**, **Steel**, **Titanium**, and **Water**. Select **Custom** to make further adjustments of the specific settings for colors, texture, reflectance, and so on. The default custom settings are inherited from the material selected last from the **Family** list.

Specular Color, Diffuse Color, and Ambient Color

For each of these properties, click the **Color** button to assign a **Custom** specular color or select a standard color from the list: **Black**, **Blue**, **Cyan**, **Gray**, **Green**, **Magenta**, **Red**, **White**, or **Yellow**.

The combination of **Specular color**, **Diffuse color**, and **Ambient color** gives a 3D object its overall color:

- **Specular color** is the color of the light of a specular reflection (specular reflection is the type of reflection that is characteristic of light reflected from a shiny surface).
- **Diffuse color** represents the true color of an object; it is perceived as the color of the object itself rather than a reflection of the light. The diffuse color gets darker as the surface points away from the light (shading). As with Ambient color, if there is a texture, this is multiplied by the colors in the texture, otherwise it is as if it has a white texture.
- **Ambient color** is the color of all the light that surrounds an object; it is the color seen when an object is in low light. This color is what the object reflects when illuminated by ambient light rather than direct light. Ambient color creates the effect of having light hit the object equally from all directions. As with Diffuse color, if there is a texture, this is multiplied by the colors in the texture; otherwise, it is as if it has a white texture.

For examples of specular, diffuse, and ambient light, which are related to these definitions, see About the 3D View Light Sources and Attributes in the *COMSOL Multiphysics Reference Manual*.

Noise

The **Noise** check box is selected by default, with the default **Normal vector noise scale** and **Normal vector noise frequency** taken from the material. Enter other values as needed, or click to clear the **Noise** check box.

- Noise is a texture that disturbs the normals when calculating lighting on the surface. This causes the surface to look rough and textured.

- Normal vector noise scale is the power of the noise texture. A high value creates a stronger texture of the surface. A value between 0–1 is suitable.
- Normal vector noise frequency is the size of the noise disturbances. A small value creates smaller features on the texture. A value between 0–10 is suitable.

Diffuse and Ambient Color Opacity

The default **Diffuse and ambient color opacity** is 1.

Lighting Model

The default **Lighting model** — **Blinn-Phong** or **Cook-Torrance** — is based on the material. Select **Simple** instead as needed.

The different lighting models provide a set of techniques used to calculate the reflection of light from surfaces to create the appropriate shading. For example, a specular highlight is the bright spot of light that appears on shiny objects when illuminated. Specular highlights are important in 3D computer graphics because they provide a strong visual cue for the shape of an object and its location with respect to light sources in the scene.

For **Blinn-Phong**, the default **Specular exponent** is 64. The specular exponent determines the size of the specular highlight. Typical values for this property range from 1 to 500, with normal objects having values in the range 5 to 20. This model is particularly useful for representing shiny materials.

For **Cook-Torrance**, the default **Reflectance at normal incidence** and **Surface roughness** are taken from the material. The Cook-Torrance lighting model accounts for wavelength and color shifting and is a general model for rough surfaces. It is targeted at metals and plastics, although it can also represent many other materials.

- Reflectance at normal incidence is the amount of incoming light (0–1) from the normal direction (of the surface) that is reflected.
- Surface roughness is a value that describes microreflectance on the surface. Higher values create a rougher look of the surface with fewer highlights. A value from 0–1 is suitable.

Property Groups

The **Settings** window for **Property Group** is where output properties and model inputs are added, local properties are defined, and expressions for material properties are entered in a specific property group such as **Basic**. The property groups are subnodes to a material node. The **Settings** window for **Property Group** is displayed when you click the property group node (for example, **Basic**) under the material node (typically with the material's name — **Aluminum**, for example) in the **Model Builder**.

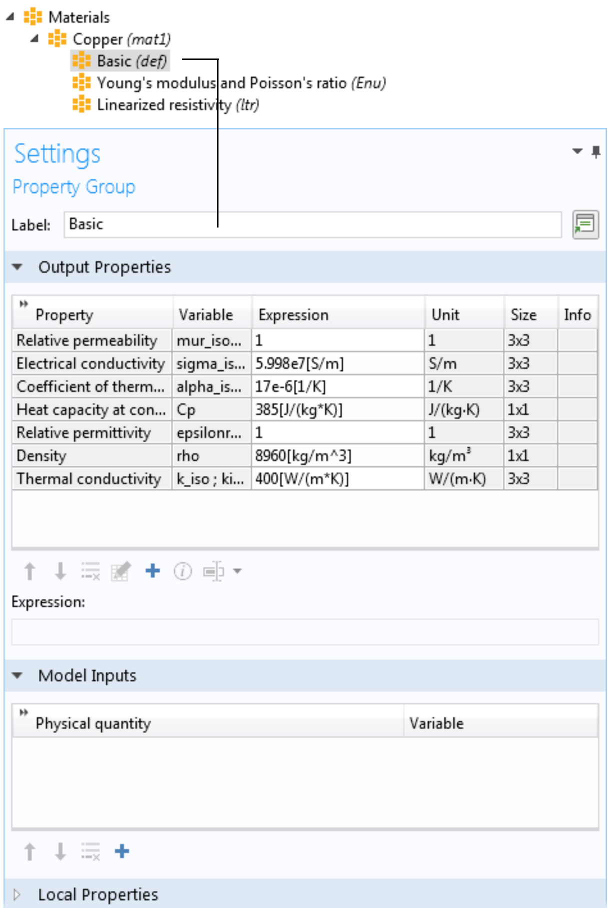


Figure 2-5: An example of a Basic Settings window for Property Group.

A property group under a material creates the following variables:

TABLE 2-2: VARIABLES GENERATED FROM A PROPERTY GROUP

TYPE	VARIABLE NAME	SCOPE	SELECTION	EXAMPLE
Basic property	Variable name of physical quantity	`root.material`	Material selection	`root.material.rho`
		`root.<comp>.<mat>.<group>`	Global selection	`root.comp1.mat1.def.rho`
Output property	Property name	`root.material.group`	Material selection	`root.material.linzRes.alpha`
		`root.<comp>.<mat>.<group>`	Global selection	`root.comp1.mat1.linzRes.alpha`

OUTPUT PROPERTIES

The predefined material properties in the property group appear in a table in the **Output Properties** section.

> It is only possible to add, move, and delete output properties from the **Basic** material properties and with user-defined property groups.

Click the **Add** button (+) to add another output property, which you choose from one of the available physical quantities in the **Physical Quantity** dialog box that opens.

If required, edit the expressions in the list's **Expression** column. Edit directly in the table or in the **Expression** field underneath the table. You can insert predefined expressions by clicking the **Insert Expression** button ( ) or clicking Ctrl+Space and then choosing an expression from the list of predefined expressions. You can also click the **Edit** button ( ), which opens a dialog box for easier specification of orthotropic and anisotropic material properties (tensors), when applicable. Select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** when entering the data in the material property's dialog box. In the **Expression** column, use a syntax with curly braces such as `{k11, k21, k31, k12, k22, k32, k13, k23, k33}` to enter anisotropic material properties for a 3-by-3 tensor k_{ij} in the order k_{11}, k_{21}, k_{31}, k_{12}, k_{22}, k_{32}, k_{13}, k_{23}, and k_{33}. 1, 2, and 3 represent the first, second, and third direction in the active coordinate system. In many cases (for example, when entering the elasticity matrix for structural mechanics), the matrix must for physical reasons be symmetric. The upper diagonal part of the matrix you enter will then be mirrored when forming the actual constitutive matrix, and the lower diagonal part is ignored.

The **Variable** column lists the variable names depending on the type of anisotropy. For an isotropic k, `k_iso` represents its single scalar value.

The **Unit** and **Size** columns provide information about the unit and size of the output property. The size is 1x1 for a scalar value such as density and 3x3 for a tensor (matrix) quantity such as electrical conductivity.

If desired, you can add information about the property, such as references for its value or expression. To do so, click the **Edit/Show Property Information** button ( ) and enter the property information in the dialog box that opens and then click **OK**. When information is available for a property, and information symbol ( ) appears in the **Info** column.

Use the **Move up** ( ), **Move down** ( ), and **Delete** ( ) buttons to organize the table as needed.

MODEL INPUTS

The model inputs are physical quantities, such as temperature, that are used as inputs in the expressions that define the output properties (for example, to describe a temperature-dependent physical quantity). For example, adding **Temperature** as a model input with the variable name `T` makes it possible to use an expression for the heat capacity at constant pressure C_p, such as `300[J/(kg*K)]*T[1/K]`, which works regardless of the name of the actual dependent variable for temperature in the model that uses the temperature-dependent material. Without the model input, the expression above only works with a temperature variable called `T`.

Click the **Add** button ( ) to add another model input, which you choose from one of the available physical quantities in the **Physical Quantity** dialog box that opens.

Use the **Move up** ( ), **Move down** ( ), and **Delete** ( ) buttons to organize the table as needed.

LOCAL PROPERTIES

Here you can enter a user-defined property by entering its variable name in the **Name** column and its corresponding **Expression** and organizing the table as needed. You can also enter a **Description**, which appears in the **Property** column in the **Material Contents** section of the parent **Material** node. In that node, the **Name** entered here appears in the **Variable** column. These local properties are useful for parameterizing functions that describe material properties if they contain inputs other than those that are model inputs (such as temperature and pressure). For example, a local property can be a reference value at a certain temperature. Use the **Move up** ( ), **Move down** ( ), and **Delete** ( ) buttons to organize the tables as needed.

You can use local properties to parameterize a material (for example, to create a generic “template” material for a particular symmetry class of anisotropic materials). You can then adjust the local property values for each instance of the material.

About Automatic Adding of Property Groups to a Material

Material property groups are automatically added to the material node in the **Model Builder**. You can also add additional predefined property groups or create a **User-Defined Property Group** (on the **Materials** toolbar, click **User-defined Property Group** ( ) or right-click the **Material** node). The available properties are collected in property groups according to the physical context.

Each property group has a **Settings** window for **Property Group**. When a **Model Builder** node is clicked (for example, **Basic**), the **Settings** window for **Property Group** displays specific information about that property group. The physical properties for all property groups are summarized in a **Material Contents** table on the **Settings** window for the parent **Material** node.

Material Link

Add a **Material Link** node ( ) under a **Materials** node in a model component to add a link to a material that you have added under the global **Materials** node ( ) and use it as a material in that component’s geometry. The **Material Link** node’s **Settings** window is similar to the **Settings** window for a material node (see The Settings Window for Material), with the exception that there is no **Material Properties** sections. Instead, it includes the following section:

LINK SETTINGS

From the **Material** list, select the global material that you want to link to:

- Any global material node, to use that material in the component.
- Any **Switch** node, if you want to run a material sweep.
- **None**, to not link to any global material.

Click the **Go to Material** button ( ) to move to the selected material node. Click the **Add Material from Library** button ( ) to add a global material from the material libraries or a new blank global material. The added material then becomes the one selected in the **Material** list.

Switch for Materials

Use the **Switch** node ( ) to switch between materials during a solver sweep. You add the materials as subnodes under the **Switch** node. Right-click to add a **Blank Material** or select **Add Material from Library** to select materials from the libraries in the **Add Material** window. The switch for materials acts essentially as a switch statement in a programming language; that is, it dynamically selects one of its underlying branches depending on a parameter that can be controlled from the solvers, using a Material Sweep study.

The **Switch** node's **Settings** window contains the following sections:

MATERIAL CONTENTS

This section lists all of the material properties that are defined for the material or required by the physics in the model on domains where the **Switch** node is the active domain material. The table lists the **Property**, **Name**, **Value**, and **Unit** for the material property as well as the **Property group** to which the material property belongs. The **Property group** corresponds to the subnodes in the **Model Builder** with the same name. If required, edit the values or expression for the property's **Value**.

The list includes properties that are defined by any of the materials under the **Switch** node. The left column provides visual cues about the status of each property:

- A stop sign ( ) indicates that some subnode is missing a required **Value**. That is, the material property is required by a physics feature in the model but is not defined for all switch cases.
- A warning sign ( ) indicates that the material property has been added to some material subnode but is still undefined.
- A green check mark ( ) indicates that the property has a **Value** in all subnodes and is currently being used in the physics of the model.

APPEARANCE

The settings in this section make it possible to control or change the default appearance of the material switch in the Graphics window when working in the materials or physics parts of the model tree. See The Settings Window for Material for more information.

Layered Material

In the **Layered Material** node ( ), you can specify the properties of a multilayer laminate. It is used when defining the properties of the following features:

- The Layered Shell interface (requires the Composite Materials Module).
- **Layered Linear Elastic Material** in the Shell interface (requires the Composite Materials Module).
- **Thin Layer** in the Heat Transfer in Solids interface.
- The Heat Transfer in Shells interface (requires the Heat Transfer Module).
- The Electric Currents, Layered Shell interface (requires the AC/DC Module).

A **Layered Material** node can be present in two locations in the Model Builder:

- The most common place is under **Global Definitions>Materials**. When you reference a layered material from a physics interface, you do it indirectly through either a Layered Material Link or a Layered Material Link (Subnode) under **Materials** in the current component.
- It can also be a subnode under a Layered Material Stack node in a component.

LAYER DEFINITION

In this table you specify the properties of each layer.

Click the **Add** button ( ) to add another table row. Use the **Move up** ( ), **Move down** ( ), and **Delete** ( ) buttons to organize the table as needed. To completely reset the table to its default state, you can use the **Reset to Default** button ( ).

Conceptually, the layers are ordered from bottom to top of the laminate. Enter the following data in the table:

Layer

Here you can assign a name to the layer for future reference. The default is a sequential numbering: Layer 1, Layer 2, and so on.

Material

Select any available material. If the **Layered Material** node is located under **Global Definitions**, the list contains only global materials. If the **Layered Material** node is used as a subnode to a **Layered Material Stack**, also materials defined under **Materials** in the component are available.

When you have a certain row in the table selected, you can access three shortcuts:

- Click the **Blank Material** ( ) button to add a new blank material under global materials. The material is referenced in current row of the **Material** column.
- Click the **Add Material from Library** ( ) button to add a new material under global materials from Material Libraries. The material is referenced in current row of the **Material** column.
- Click the **Go to Material** ( ) button to jump to the definition of the material selected on the current row.

When you add a new row to the table, the same material as on the previous row is selected. This means that if you have many, not adjacent, layers with the same material, it is more efficient to initially add all layers with that same material. Then you can go back and change the material for some layers. Alternatively, you can reorder the layers using the **Move up** ( ) and **Move down** ( ) buttons.

Rotation

If the material in the layer is orthotropic or anisotropic, enter the angle in degrees (positive counterclockwise) from the first principal axis of the laminate to the first principal axis of the layer. Even for an isotropic material, the orientation can matter for result presentation, since it affects the interpretation of for example stress tensor components.

Thickness

Enter the thickness of the layer (default unit: m).

Mesh elements

In the physics interfaces, the layered materials are handled through the concept of a virtual *extra dimension*. For a layered material defined on a boundary, you can think of that as an extra coordinate in the normal direction. Enter the number of elements that you want in the extra dimension for the layer.

INTERFACE PROPERTY

In some physics features, not only the layers themselves but also the interfaces between them are important. In such a case, you can assign materials to the interfaces in this table. The number of interfaces is one more than the number of layers because the free top and bottom surfaces of the laminate are also considered as interfaces.

In most cases, you do not need to enter anything in this section.

Interface

This is the interface name, for future reference. As a default, the interface name is constructed from the names of the two adjacent layers. For the top and bottom interfaces, the labels “up” and “down” are used for the two exterior sides.

You can rename the interfaces. This is, however, seldom needed.

Position

This column shows the location of the interface. The distance is counted from the bottom of the laminate. The column is for information only, and cannot be modified.

Material

Select the material of the interface. You only need to assign materials to the interfaces which are explicitly referenced by physics features.

Figure 2-6 shows an example of the settings for a layered material. The layer names have been entered manually, whereas the interfaces have retained their default names.

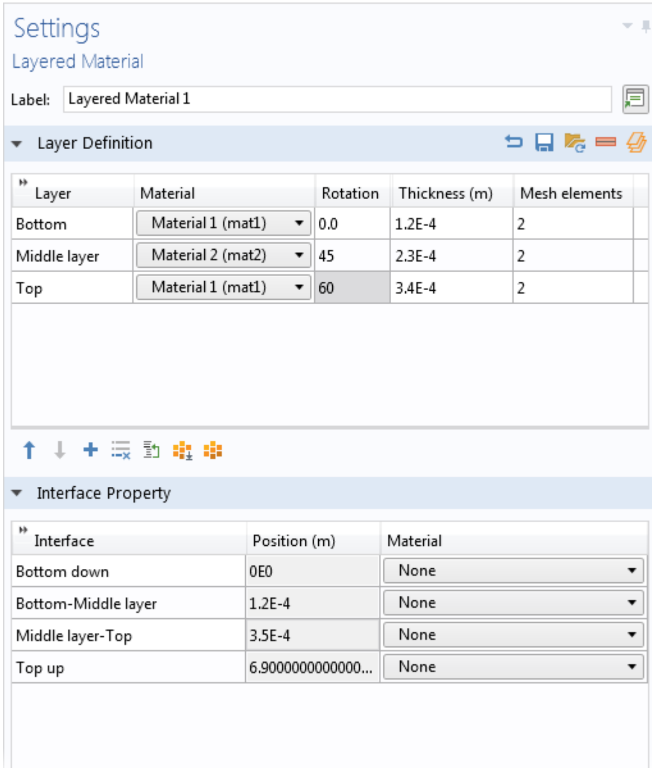


Figure 2-6: Settings for a material with three layers.

You can save the laminate definition to a text file by clicking the **Save Layers to File** ( ) button. For the example above, the text file has the following contents:

```
Bottom mat1 0.0 1.2E-4 2
"Middle layer" mat2 45 2.3E-4 2
Top mat1 60 3.4E-4 2
```

To load a text file on this format, click the **Load Layers from File** ( ) button. For complex laminates, it may be easier to start by creating the text file representation in a text editor, than to enter the data in the GUI.

When loading a file, the second column containing the material tag is ignored. The reason is that there is no way to ascertain that a material tag like 'mat2' would point to the same material in another context. You can even load a file where that column is absent.

You have two options for visualizing the laminate defined in the **Layered Material** node. To see the thickness of each layer, click the **Layer Cross Section Preview** ( ) button. This will give a plot like the one shown in Figure 2-7.

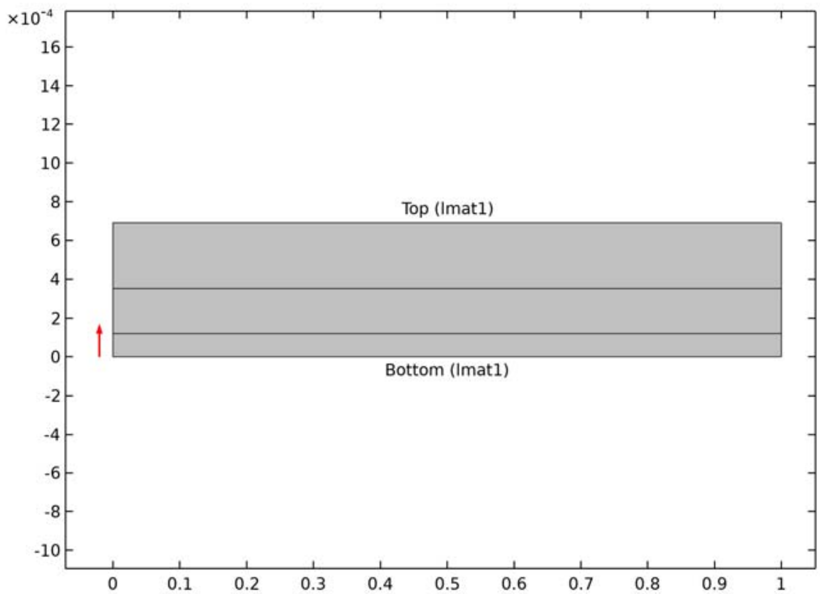


Figure 2-7: The layer cross section plot for a material with three layers.

To visualize the layer orientations, click the **Layer Stack Preview** ( ) button. In Figure 2-8, an example of such a plot is shown. The x-axis corresponds to the principal laminate direction, and the stripes indicate the principal direction of each layer.

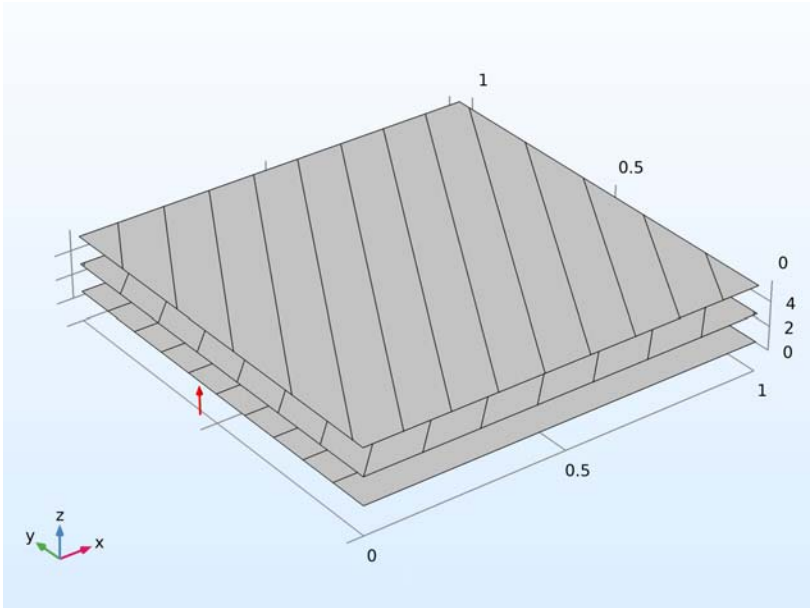


Figure 2-8: The layer stack preview plot for a material with three layers.

PREVIEW PLOT SETTINGS

In this section, you can fine-tune the display in the preview plots.

In the **Distance between the orientation lines** text field, you can enter a value for the spacing of the stripes showing the orientation of the principal orientation of the layer. The layer itself is always drawn as a square with the unity side length. If you deselect the corresponding check box, no orientation lines are drawn.

The value of the **Thickness-to-width ratio** is used by both types of preview plots.

- In a layer stack preview plot, it controls the height of the stack in the *z*-direction. For laminates with many layers, you may need to increase this value.
- In the layer cross section preview plot, it controls the height in the *y*-direction. The width is always unity.

Clear the **Shows labels in cross section plot** check box to remove the text labels showing layer names and materials.

Layered Material Link

The **Layered Material Link** node (icon) provides a bridge from a Layered Material, located under **Global Definitions**, to a physics feature residing in a component. A physics feature designed to work with layered materials cannot directly reference a **Layered Material**. The **Layered Material Link** node is located in the **Layers** submenu under a **Materials** node.

ORIENTATION AND POSITION

Select a **Coordinate system** defining the principal directions of the laminate. The orientation of each layer, given in the **Layered Material** node, is a rotation from the first coordinate axis of this coordinate system. Only **Boundary System** coordinate systems can be selected.

Choose a **Position** — **Midplane on boundary**, **Down side on boundary**, **Up side on boundary**, or **User defined**. This controls the possible offset of the layered material from the geometrical boundary on which the mesh exists (the *reference surface*). For **User defined**, enter a value for the **Relative midplane offset**. The value 1 corresponds to **Down side on boundary**, and the value -1 corresponds to **Up side on boundary**. Values may be outside the range -1 to 1, in which case the reference surface is outside the laminate.

The **Position** setting is only used by physics features where the physical behavior depends of the actual location, such as structural shells.

By clicking the **Layer Cross Section Preview** (icon) button, you get a preview plot of the layered material, including the location of the reference surface (Figure 2-9). The height of the laminate in the plot is controlled by the value of the **Thickness-to-width ratio** specified in the **Preview Plot Settings** for the selected layered material.

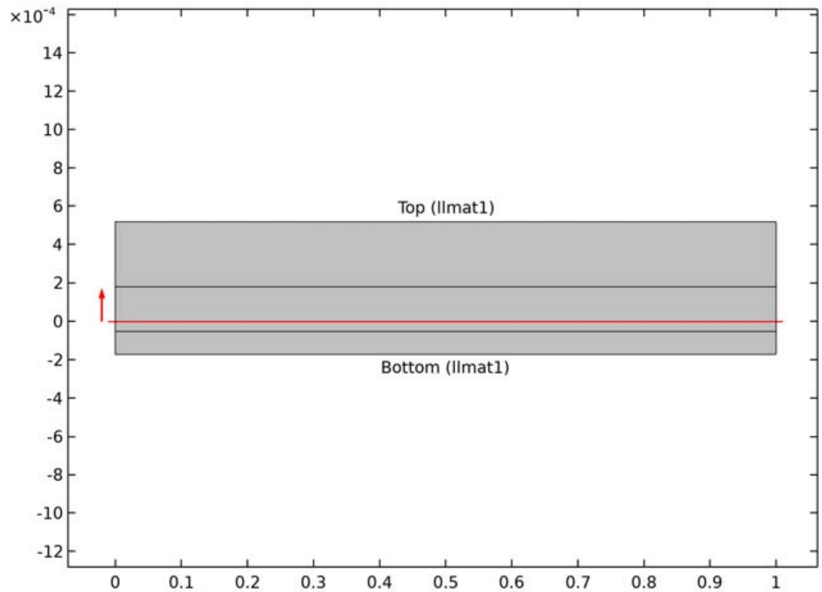


Figure 2-9: Layer cross section preview plot with relative offset set to 0.5.

NONLAYERED MATERIAL SETTING

In some cases, a single standard material definition is needed on the same boundary as a layered material. This can, for example, be the case if two different physics interfaces are active on the same boundary, but only one of them supports a layered material definition. You can select any nonlayered material from the **Material** list. This selection is completely analogous to using a Material Link.

> You cannot use an ordinary **Material** or **Material Link** with the same selection as the **Layered Material Link**. These nodes override each other.

By clicking the **Go to Material** ( ) button, you can jump to the settings for the selected material.

Click the **Add Material from Library** button ( ) to add a global material from the material libraries or a new blank global material. The added material then becomes the one selected in the **Material** list.

LAYERED MATERIAL SETTING

Select a layered material from the **Material** list. You can also select a Switch for Materials.

By clicking the **Go to Material** ( ) button, you can jump to the settings for the selected material.

Click the **Add Layered Material** button ( ) to add another **Layered Material** or a **Switch**. The added material then becomes the one selected in the **Material** list.

MATERIAL CONTENTS

See the documentation for Material Contents for the **Material** node.

The **Value** column will usually contain the string `Layer`, indicating that the actual value is layer dependent.

APPEARANCE

See the documentation for Appearance for the **Material** node.

Layered Material Stack

In the **Layered Material Stack** node ( ), you can compose a new layered material by stacking other layered materials on top of each other. There are three main reasons why you may want to do this:

- The layup is repetitive, say with the same four layers repeated five times. Rather than defining twenty layers in a **Layered Material** node, you define four, and then add this definition five times in a **Layered Material Stack**.
- There are layer drop-offs, that is some layers are not present everywhere in the structure. Then, it is efficient to create only subsets of the laminate in **Layered Material** nodes, and use a number of **Layered Material Stack** nodes to combine them into different configurations.
- Two **Layered Material Stack** nodes can have parts of their definitions linked to the same **Layered Material** node. When a transition through a continuity feature is used, the corresponding layers in two laminates defined as stacks can be connected automatically.

The **Layered Material Stack** node is located in the Layers submenu under a Materials node.To compose the stack, you add subnodes to the **Layered Material Stack**. These subnodes can be either a Layered Material or a Layered Material Link (Subnode). You can add any number of subnodes, and mix the two types. The order of the subnodes determines the ordering of the layers in the final laminate.

ORIENTATION AND POSITION

Select a **Coordinate system** defining the principal directions of the laminate. The orientation of each layer, given in the **Layered Material** node, is a rotation from the first coordinate axis of this coordinate system. Only **Boundary System** coordinate systems can be selected.

Choose a **Position** — **Midplane on boundary**, **Down side on boundary**, **Up side on boundary**, or **User defined**. This controls the possible offset of the layered material from the geometrical boundary on which the mesh exists (the *reference surface*). For **User defined**, enter a value for the **Relative midplane offset**. The value 1 corresponds to **Down side on boundary**, and the value –1 corresponds to **Up side on boundary**. Values may be outside the range –1 to 1, in which case the reference surface is outside the laminate.

The **Position** setting is only used by physics features where the physical behavior depends of the actual location, such as structural shells.

By clicking the **Layer Cross Section Preview** ( ) button, you get a preview plot of the stacked layered material, including the location of the reference surface. In Figure 2-10, a laminate composed of three stacked layered materials, each consisting of three layers is shown. Note that there is a slight indentation, used for emphasizing the transition from one part of the stack to the next.

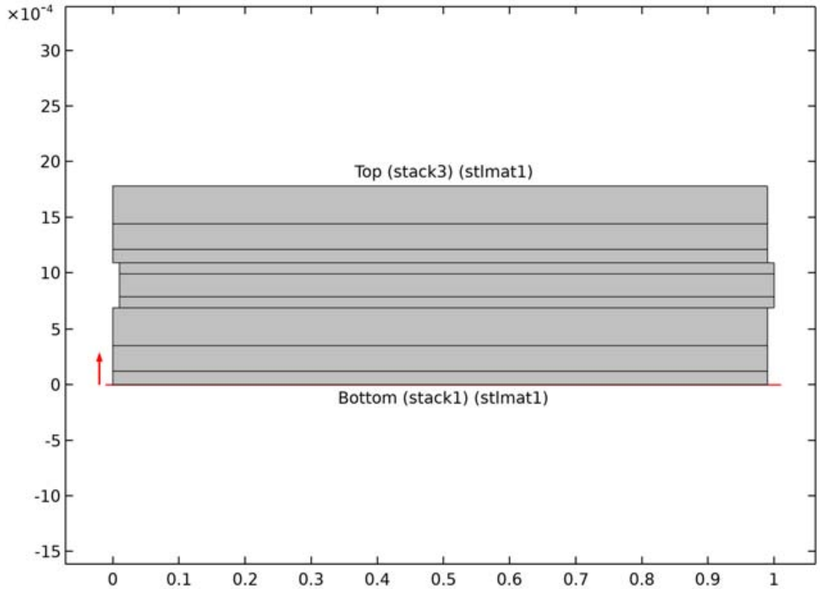


Figure 2-10: Layer cross section preview plot with relative offset set to Down side on boundary.

NONLAYERED MATERIAL SETTING

In some cases, a single standard material definition is needed on the same boundary as a layered material. This can for example be the case if two different physics interfaces are active on the same boundary, but only one of them supports a layered material definition. You can select any non-layered material from the **Material** list. This selection is completely analogous to using a Material Link.

> You cannot use an ordinary **Material** or **Material Link** with the same selection as the **Layered Material Stack**. These nodes override each other.

By clicking the **Go to Material** ( ) button, you can jump to the settings for the selected material.

Click the **Add Material from Library** button ( ) to add a global material from the material libraries or a new blank global material. The added material then becomes the one selected in the **Material** list.

PREVIEW PLOT SETTINGS

In this section, you can fine-tune the display in the preview plot.

The value of the **Thickness-to-width ratio** controls the height in the y-direction. The width is always unity.

Deselect the **Shows labels in cross section plot** check box to remove the text labels showing layer names and materials.

MATERIAL CONTENTS

See the documentation for Material Contents for the **Material** node.

The **Value** column will usually contain the string `Layer`, indicating that the actual value is layer dependent.

APPEARANCE

See the documentation for Appearance for the **Material** node.

Layered Material Link (Subnode)

The **Layered Material Link** subnode ( ) is used for referencing a Layered Material from a Layered Material Stack node. You can add any number of **Layered Material Link** subnodes under a **Layered Material Stack** node.

LINK SETTINGS

Select a layered material from the **Material** list.

By clicking the **Go to Material** ( ) button you can jump to the settings for the selected material.

Click the **Add Layered Material** button ( ) to add another **Layered Material** or a **Switch**. The added material then becomes the one selected in the **Material** list.

Single Layer Material

The **Single Layer Material** node ( ) provides a quick way to define data for a non-layered material to be used in physics feature designed for layered materials. Using a single layer material is equivalent to defining a **Layered Material** with only one layer, and then referencing it through a **Layered Material Link**.

LAYER DEFINITION

- Enter a **Label**, which can be used for future reference. The default is `Layer`.

- Enter the layer **Thickness** (default unit: m)
- Enter the **Rotation**. If the material in the layer is orthotropic or anisotropic, enter the angle in degrees (positive counterclockwise) from the first principal axis of the laminate to the first principal axis of the layer. Even for an isotropic material, the orientation can matter for result presentation, since it affects the interpretation of for example stress tensor components.
- Select a **Material** from the list. It contains all global materials, as well as all materials defined in the current component.

  By clicking the **Go to Material** ( ) button you can jump to the settings for the selected material.

  Click the **Add Material from Library** button ( ) to add a global material from the material libraries or a new blank global material. The added material then becomes the one selected in the **Material** list.
- Enter the number of **Mesh elements**. In the physics interfaces, the layered materials are handled through the concept of a virtual *extra dimension*. For a layered material defined on a boundary, you can think of this as an extra coordinate in the normal direction. Enter the number of elements that you want in the extra dimension for the layer.

INTERFACE PROPERTY

In some physics features, not only the layers themselves but also the interfaces between them are important. In such a case, you can assign materials to the interfaces in this table. The single layer shell has two external interfaces — the downside and the upside — and no internal interfaces. You can specify data for the two sides individually.

In most cases, you do not need to enter anything in this section.

The **Label** is the interface name, for future reference. As a default, the interface names are `Downside` and `Upside`, respectively.

Select the **Material** of each interface.

By clicking the **Go to Material** ( ) button, you can jump to the settings for the selected material.

Click the **Add Material from Library** button ( ) to add a global material from the material libraries or a new blank global material. The added material then becomes the one selected in the **Material** list.

NONLAYERED MATERIAL SETTING

In some cases, a single standard material definition is needed on the same boundary as a layered material. This can for example be the case if two different physics interfaces are active on the same boundary, but only one of them supports a layered material definition. You can select any non-layered material from the **Material** list. This selection is completely analogous to using a Material Link.

> You cannot use an ordinary **Material** or **Material Link** with the same selection as the **Single Layered Material**. These nodes override each other.

By clicking the **Go to Material** ( ) button, you can jump to the settings for the selected material.

Click the **Add Material from Library** button ( ) to add a global material from the material libraries or a new blank global material. The added material then becomes the one selected in the **Material** list.

ORIENTATION AND POSITION

Select a **Coordinate system** defining the principal directions of the laminate. The orientation of the single layer, as given in **Rotation**, is a rotation from the first coordinate axis of this coordinate system. Only **Boundary System** coordinate systems can be selected.

Choose a **Position** — **Midplane on boundary**, **Down side on boundary**, **Up side on boundary**, or **User defined**. This controls the possible offset of the material from the geometrical boundary on which the mesh exists (the *reference surface*). For **User defined**, enter a value for the **Relative midplane offset**. The value 1 corresponds to **Down side on boundary**, and the value –1 corresponds to **Up side on boundary**. Values may be outside the range –1 to 1, in which case the reference surface is outside the laminate.

The **Position** setting is only used by physics features where the physical behavior depends of the actual location, such as structural shells.

By clicking the **Layer Cross Section Preview** ( ) button, you get a preview plot of the single layer material, including the location of the reference surface. This plot looks similar to Figure 2-9, but there is only a single layer.

PREVIEW PLOT SETTINGS

In this section, you can fine-tune the display in the preview plot.

The value of the **Thickness-to-width ratio** controls the height in the y-direction. The width is always unity.

Deselect the **Shows labels in cross section plot** check box to remove the text labels showing layer names and materials.

MATERIAL CONTENTS

See the documentation for Material Contents for the **Material** node.

The **Value** column usually contains the string `Layer`, indicating that the actual value is layer dependent.

Material Properties

The materials included in the Material Library are defined by unique material properties, each available as a function of temperature or another appropriate argument. Table 2-4 lists most of the material properties in the Material Library.

!	It is important to check the validity of the material property function under the conditions that you are interested in investigating. See Checking the Validity of Properties in the Material Library.

	Individual material properties contained in the Material Library are based on the Material Property Database (MPDB) from JAHM Software, Inc.

Viewing Material Property Information

For all properties contained in the Material Library, you can view applicable literature references, notes, and reference temperatures in the Material Browser's **Property reference** section.

1 Open the Material Browser.

2 Under **Material Library**, click to select a material. For example, **Nitrogen**. The information about this material displays on the right-hand side of the window.

3 Under **Properties** in the table, click a **Property** to see its references in the **Property reference** section. See Figure 2-11.

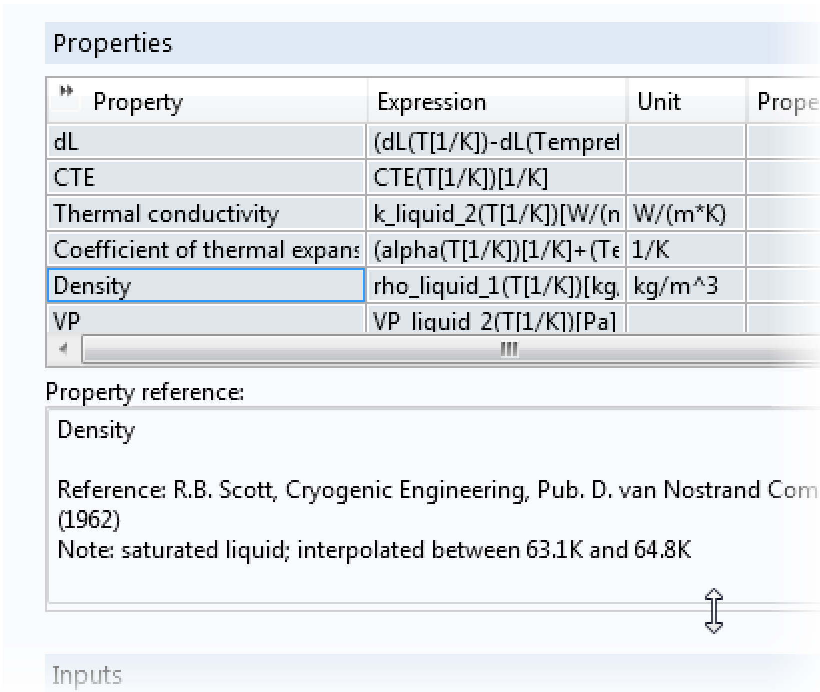


Figure 2-11: An example of where you can find the property reference information for a material. In this example, Density has this information available in the Property reference section. You can hover over the section and drag to expand it if required.

Functions Default Values in the Material Library

The material property expressions stored in the Material Library contain calls to the corresponding material property functions using input variables (arguments) as in Table 2-3. The default variable name can be changed in the property expressions to match actual variable names in a model. When a material property is used by a physics feature set to retrieve the property **From material** this is not necessary. In that case, the actual variable are retrieved from the **Model Inputs** section of the same feature and automatically substituted into the material property expression.

If you, on the other hand, want to access material properties from a material explicitly, you need to make sure that the function argument variables used in the property expressions exist and can be evaluated in the model. For example, if the variable `T2` is used for temperature, change the argument of the property functions from T to T2 in the expressions.

The argument does not have to be a variable defined by the model (such as dependent variables) — it can also be a user-defined constant or variable. In general, COMSOL Multiphysics tries to find the best match for evaluating function arguments when material properties are accessed explicitly.

TABLE 2-3: DEFAULT FUNCTION ARGUMENTS IN MATERIAL PROPERTY EXPRESSIONS

ARGUMENT	DEFAULT VARIABLE	UNIT
Temperature	T	K
Time	t	h
Effective plastic strain	epe	-
Number of cycles	n	-
Norm of H field	normH_emnc	A/m
Norm of B field	normB_emqa	T

Available Material Library Material Properties

The following table lists the material properties in the Material Library:

TABLE 2-4: MATERIAL LIBRARY: AVAILABLE MATERIAL PROPERTIES

PROPERTY	SHORT NAME	ARGUMENT	SI UNIT
Coefficient of thermal expansion	alpha	Temperature	1/K
Creep strength	CS	Time	Pa
Density	rho	Temperature	kg/m^3
Dynamic viscosity	eta	Temperature	Pa·s
Electrical conductivity	sigma	Temperature	S/m
Resistivity	res	Temperature	ohm·m
Elongation	elong	Temperature	-
Fatigue E-N curve	FEN	Number of cycles	Pa
Fatigue S-N curve	FSN	Number of cycles	Pa
Heat capacity	C	Temperature	J/(kg·K)
Bulk modulus	K	Temperature	Pa
Shear modulus	G	Temperature	Pa

TABLE 2-4: MATERIAL LIBRARY: AVAILABLE MATERIAL PROPERTIES

PROPERTY	SHORT NAME	ARGUMENT	SI UNIT
Instantaneous coefficient of thermal expansion	CTE	Temperature	1/K
Linear expansion	dL	Temperature	-
Molar heat capacity	HC	Temperature	J/(mol·K)
Nonlinear magnetic flux density, norm	normB	Norm of H field	T
Nonlinear magnetic field, norm	normH	Norm of B field	A/m
Normal total emissivity	nemiss	Temperature	-
Poisson’s ratio	nu	Temperature	-
Relative permeability	mur	Norm of H field	-
Stress rupture	SR	Time	Pa
Surface emissivity	epsilon	Temperature	-
Tensile strength	Syt	Temperature	Pa
Thermal conductivity	k	Temperature	W/(m·K)
Thermal diffusivity	TD	Temperature	m^2/s
True stress–true strain curve in tension	Syfunc	Strain	Pa
True stress–true strain curve in compression	Syfunccomp	Strain	Pa
Vapor pressure	VP	Temperature	Pa
Yield strength level	Sys	Temperature	Pa
Young’s modulus	E	Temperature	Pa

Checking the Validity of Properties in the Material Library

The following section lists points to consider about the definition, error estimate, and conditions for some of the Material Library properties listed in Table 2-4.

The property functions listed below have a literature reference where you can find more details about the conditions and validity range for that specific property.

COEFFICIENT OF THERMAL EXPANSION

- The coefficient is defined as $(\Delta L/L)_T/(T - T_{\text{ref}})$ and in most cases, it is calculated from the $\Delta L/L$ values.
- The error is expected to be in the range of 10–15%, but it might be higher near room temperature due to the small value of $T - T_{\text{ref}}$.

ELASTIC AND INITIAL SHEAR MODULUS

- The data accuracy is approximately 5–10%.
- For solder alloys the literature reports a wide spread of values. Data from several sources (when available) are evaluated, and representative values are given; the error is estimated to be 10–25%.
- For some polymers the flexural modulus is used as the elastic modulus, and it is typically within 10% of the elastic modulus.
- Typically, values measured with a strain gauge are approximately 10% lower than those measured with a dynamic technique.
- Values measured by a dynamic technique are preferred over those measured by strain gauge techniques.
- For cubic materials where the elastic and shear modulus are calculated from the elastic constants (C11, C12, and C44), the Material Library uses the average of the Reuss and Voigt equations (see R.F.S. Hearmon, *Advances in Physics*, vol. 5, 1956, p. 232).
- For isotropic solids (glasses), it uses methods from L.D. Landau and E.M. Lifshitz, *Theory of Elasticity*, Addison-Wesley, New York, 1966.

POISSON'S RATIO AND INITIAL BULK MODULUS

- Calculated from the elastic modulus and the shear modulus using standard relationships, and in this sense they are self-consistent and accurate.
- Data accuracy is approximately 10–20%. Because these are derived quantities the error can be significantly higher.
- The curves for these properties often show improbable shapes that are most likely due to their derived nature and are not believed to be real. If the elastic and shear modulus were determined in a self-consistent manner, the curves would likely be much better behaved. However, all of the data are presented “as is” from the original references and are self-consistent within the Material Library.

THERMAL CONDUCTIVITY

- Can be very sensitive to impurities, heat treatment, and mechanical worked state, especially at very low temperatures.
- The sensitivity is somewhat decreased above room temperature and decreases as the amount of alloying increases. Compare 4340-QT (quenched and tempered) and 4340-NT (annealed).

THERMAL DIFFUSIVITY

- For metals this property can be very sensitive to impurities, heat treatment, and mechanical worked state, especially at very low temperatures.
- This sensitivity is somewhat decreased above room temperature and decreases as the amount of alloying increases. To see an example of this, compare the data for elemental (high purity) Fe and Armco iron (commercial purity).

ELECTRIC RESISTIVITY

This property is very sensitive to impurities, heat treatment, and mechanical worked state, especially at very low temperatures.

ELECTRICAL CONDUCTIVITY

This property is very sensitive to impurities, heat treatment, and mechanical worked state, especially at very low temperatures.

SURFACE EMISSIVITY (ε_T)

This property is the measured emissivity over all wavelengths and 2π radians. This is the emissivity used in the Stefan-Boltzmann law.

NORMAL TOTAL EMISSIVITY ($\varepsilon_{T,n}$)

- The measured emissivity is over all wavelengths at a direction normal to the surface. This is the most commonly reported value.
- For polished metal, this assumption is valid: $\varepsilon_T/\varepsilon_{T,n} = 1.15–1.20$.
- Both emissivities are sensitive to the surface condition (roughness and oxide thickness).

DENSITY (ρ)

- The density for solids is calculated from the room-temperature density and the linear expansion coefficient and is given by $\rho/(1 + \Delta L/L)^3$.
- The data for oxides, carbides, and nitrides depend on the material's porosity.
- For gases the ideal gas law is used.

TENSILE STRENGTH, YIELD STRENGTH LEVEL, AND ELONGATION

Most of the data for tensile strength, yield strength level, and elongation is from supplier product brochures. When using this data, remember it is only representative of the actual material properties.

- The variation with temperature is usually not smooth. Many of these materials are precipitation hardening alloys, and the temperature affects the aging processes in different ways at different temperatures.
- Unless otherwise stated, the data are for "short" times at the indicated temperatures and not for the equilibrium structure.
- These properties are very sensitive to the details of the processing and heat treatments. Comparison of data from different suppliers indicate that the spread in the published values is approximately 20% for materials with similar processing. The spread in the elongation data can be as high as 50–100%.

FATIGUE S-N CURVE

- Fatigue data is given as the maximum stress, σ_{max}, as function of the number of cycles. The stress amplitude, maximum stress, and minimum stress are related through the stress ratio, R.

$$\sigma_a = \frac{(\sigma_{max} - \sigma_{min})}{2} \qquad R = \frac{\sigma_{max}}{\sigma_{min}}$$

- The maximum stress, σ_{max}, is given together with the stress ratio for all fatigue data. Then calculate the stress amplitude as:

$$\sigma_a = \frac{\sigma_{max}\left(1 - \frac{1}{R}\right)}{2}$$

CREEP STRENGTH AND STRESS-RUPTURE CURVES

This property is very sensitive to the test atmosphere as well as the microstructure and heat treatment of the material.

POLYMERS AND POLYMER-BASED COMPOSITES

Properties of polymers and polymer-based composites are sensitive to moisture and processing conditions, and they can show time-dependence at higher temperatures.

The errors/uncertainties can be large compared to those of other materials. Keep these aspects in mind when using the properties of these materials.

GENERAL

The magnitude of the errors reported by authors for a given property is usually smaller by a factor of 2–3 than the error between different sources for the same data. This is especially true for materials such as ceramics.

Other Material Properties Reference

In addition to the specific properties included with the Material Library, the other material databases also contain predefined variables for various material properties that can be used when creating a model.

The material properties for the predefined materials are accessible from most physics interfaces. Using this information, either create a material property group or define a completely new material.

In the **Basic>Property Group** window, you can add **Output Properties** under the **Quantities** subsection. You can also add **Model Inputs** to, for example, create a temperature- dependent material property.

About Model Inputs

Model inputs is a special type of parameter in physics features or physics properties where you can choose from a list of announced variables (typically field quantities such as temperature, concentration, or electric field, where vector fields have three components). Model inputs can also be used as an input to a **Property Group** under a material to represent, for example, a temperature-dependent material property. If the property group specifies that it supports one or more model inputs, any physics feature that uses the group's material will display those model input lists in the **Model Inputs** section of the physics node's **Settings** window. Any physical quantity in COMSOL Multiphysics can be used as a model input.

Model inputs are always available as common model inputs. See Common Model Inputs in the *COMSOL Multiphysics Reference Manual*.

All physical quantities that can act as model inputs declare and define common variables that always available (for example, `minput.T` for the temperature T).

To define the absolute pressure for heat transfer, see the settings for the Fluid node in the *COMSOL Multiphysics Reference Manual*.

To define the absolute pressure for a Fluid Flow interface, see the settings for the Fluid Properties node (described for the Laminar Flow interface in the *COMSOL Multiphysics Reference Manual*).

If you have a license for a Nonisothermal Flow interface, see that documentation for further information.

Model Inputs and Multiphysics Couplings in the *COMSOL Multiphysics Reference Manual*

About the Output Material Properties

Some of these material groups are only used by physics interfaces in the add-on modules and detailed information is in the applicable documentation.

This section describes all available property groups and the material properties that they contain. These material properties can be added to models from two **Settings** windows: the **Material** window and its subnodes' **Property Group** windows.

The Basic group contains over 25 basic properties for use with all materials.

Materials in the *COMSOL Multiphysics Reference Manual*

BASIC MATERIAL PROPERTIES

These common material properties belong to the **Basic** property group.

- When this information is accessed from the **Basic>Property Group** window, it is listed under **Quantities>Output Properties** and **Variable** is listed in the table.
- When this information is accessed from the **Material** window, it is listed under **Material Properties>Basic Properties** and **Name** is listed in the table under **Material Contents**.

TABLE 2-5: BASIC MATERIAL PROPERTIES

PROPERTY	NAME/VARIABLE	SI UNIT
Absorption Coefficient	kappaR	1/m
Activation Energy	dE	J/mol
Bulk Viscosity	muB	Pa·s
Characteristic Acoustic Impedance	Z	Pa·s/m
Coefficient of Hygroscopic Swelling	beta_h_iso, beta_h*ii*	m^3/kg
Coefficient of Thermal Expansion	alpha	1/K
Compressibility of Fluid	chif	1/Pa
Density	rho	kg/m^3
Diffusion Coefficient	D	m^2/s
Dynamic Viscosity	mu	Pa·s
Electrical Conductivity	sigma	S/m
Electron Mobility	mue	m^2/(Vs)
Extinction Coefficient	betaR	1/m
Frequency Factor	A	1/s
Heat Capacity at Constant Pressure	Cp	J/(kg·K)
Isotropic Structural Loss Factor	eta s	1
Mass Flux	Mf	kg/(m^2·s)
Mean Molar Mass	Mn	kg/mol
Permeability	kappa	m^2
Poisson’s Ratio	nu	1
Porosity	epsilon	1
Ratio of Specific Heats	gamma	1
Relative Permeability	mur	1
Relative Permittivity	epsilonr	1

TABLE 2-5: BASIC MATERIAL PROPERTIES

PROPERTY	NAME/VARIABLE	SI UNIT
Resistivity	res	Ω·m
Scattering Coefficient	sigmaS	1/m
Seebeck Coefficient	S	V/K
Shifted Magnetic Field	shiftedH	A/m
Speed of Sound	cp	m/s
Storage	S	1/Pa
Surface Emissivity	epsilon rad	1
Thermal Conductivity	k	W/(m·K)
Thermal Conductivity Supplement	b	1
Vapor Permeability	delta_p	s
Vapor Resistance Factor	mu_vrf	1
Water Content	w_c	kg/m^3
Young's Modulus	E	Pa

The coefficient of thermal expansion (CTE) and the resistivity temperature coefficient have the SI unit 1/K. COMSOL Multiphysics translates this into the Fahrenheit temperature unit using an offset. This translation means that you do not get the expected results.

Use caution when a model uses the coefficient of thermal expansion or the resistivity temperature coefficient and the unit system's temperature is not kelvin.

The rest of the material properties are grouped by application area:

- Acoustics Material Properties
- Electrochemistry Material Properties
- Electromagnetic Models
- Equilibrium Discharge
- Gas Models
- Piezoelectric Models
- Piezoresistive Models
- Solid Mechanics Material Properties
- Solid Mechanics Material Properties: Nonlinear Structural Materials Module
- Solid Mechanics Material Properties: Fatigue Module
- Solid Mechanics Material Properties: Geomechanics Material Model
- Semiconductors Material Properties

Acoustics Material Properties

Under Acoustics, you find the following acoustic material models with their associated material properties: a **Poroacoustics Model**, a **Thermoviscous Acoustics Model**, and a **Viscous Model**.

These material property groups (including their associated physical properties) can be added to models from the **Material** window. These property groups require the Acoustics Module.

TABLE 2-6: ACOUSTICS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
POROACOUSTICS MODEL		
Flow resistivity	Rf	$Pa \cdot s/m^2$
Thermal characteristic length	Lth	m
Viscous characteristic length	Lv	m
Tortuosity factor	tau	1
THERMOVISCOUS ACOUSTICS MODEL		
Bulk viscosity	muB	Pa·s
Density	rho	kg/m^3
Dynamic viscosity	mu	Pa·s
Heat capacity at constant pressure	Cp	J/(kg·K)
Thermal conductivity	k	W/(m·K)

TABLE 2-6: ACOUSTICS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
VISCOUS MODEL		
Bulk viscosity	muB	Pa·s

Electrochemistry Material Properties

These material property groups for electrochemistry (including their associated physical properties) can be added to models from the **Material** window. These property groups require the Batteries & Fuel Cells Module, Corrosion Module, Electrochemistry Module, or Electrodeposition Module.

TABLE 2-7: ELECTROCHEMISTRY MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
EQUILIBRIUM POTENTIAL		
Equilibrium potential	Eeq	V
Reference concentration	cEeqref	mol/m^3
Temperature derivative of equilibrium potential	dEeqdT	V/K
ELECTROLYTE CONDUCTIVITY		
Electrolyte conductivity	sigmal	S/m
ELECTROLYTE SALT CONCENTRATION		
Electrolyte salt concentration	cElsalt	mol/m^3
LINEARIZED RESISTIVITY	This material node defines the electric resistivity (and conductivity) as a linear function of temperature.	
Reference resistivity	rho0	Ω·m
Reference temperature	Tref	K
Resistivity temperature coefficient	alpha	1/K
OPERATIONAL ELECTRODE STATE-OF-CHARGE		
Maximum electrode state-of-charge	socmax	1
Minimum electrode state-of-charge	socmin	1
SPECIES PROPERTIES		
Transport number	transNum	1

Electromagnetic Models

These material property groups for various electromagnetic material models (including their associated physical properties) can be added to models from the **Material** window. These properties require the AC/DC Module, RF Module, or Wave Optics Module.

TABLE 2-8: ELECTROMAGNETIC MODELS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
BH CURVE	This material node is only available with the AC/DC Module.	
Local Properties	normH	-
Magnetic flux density norm	normB	T
DIELECTRIC LOSSES		
Dielectric loss factor	eta_epsilon	-
Relative permittivity (imaginary part)	epsilonBis	1
Relative permittivity (real part)	epsilonPrim	1
E-J CHARACTERISTIC	This material node is only available with the AC/DC Module.	
Electric field norm	normE	V7M
Local Properties	normJ	-
EFFECTIVE BH CURVE	This material node is only available with the AC/DC Module.	
Local Properties	normHeff	-
Magnetic flux density norm	normBeff	T
EFFECTIVE HB CURVE	This material node is only available with the AC/DC Module.	
Local Properties	normBeff	-
Magnetic field norm	normHeff	A/m
HB CURVE	This material node is only available with the AC/DC Module.	
Local Properties	normB	-
Magnetic field norm	normH	A/m
JILES-ATHERTON MODEL PARAMETERS	This material node is only available with the AC/DC Module.	
Maximum magnetization parameter	MsJA (3x3 matrix)	A/m
Langevin slope parameter	aJA (3x3 matrix)	A/m
Pinning parameter	kJA (3x3 matrix)	A/m

TABLE 2-8: ELECTROMAGNETIC MODELS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Reversibility parameter	cJA (3x3 matrix)	1
Interdomain coupling parameter	alphaJA (3x3 matrix)	1
LINEARIZED RESISTIVITY	This material node defines the electric resistivity (and conductivity) as a linear function of temperature.	
Reference resistivity	rho0	Ω·m
Reference temperature	Tref	K
Resistivity temperature coefficient	alpha	1/K
LOSS TANGENT, LOSS ANGLE	This material node assumes zero conductivity.	
Loss tangent, loss angle	delta	rad
Relative permittivity (real part)	epsilonPrim	1
LOSS TANGENT, DISSIPATION FACTOR	This material node assumes zero conductivity.	
Loss tangent, dissipation factor	tanDelta	1
Relative permittivity (real part)	epsilonPrim	1
MAGNETIC LOSSES		
Relative permeability (imaginary part)	murBis	-
Relative permeability (real part)	murPrim	-
MAGNETOSTRICTIVE		
Saturation magnetization	Ms	A/m
Initial magnetic susceptibility	chi	1
Saturation magnetostriction	lambdas	1
Magnetostriction constants	lambda100	1
Magnetostriction constants	lambda111	1
REFRACTIVE INDEX	This material node assumes a relative permeability of unity and zero conductivity. This material node is only available with the RF Module or the Wave Optics Module.	
Refractive index, imaginary part	ki	-
Refractive index	n	1
STRAIN-MAGNETIZATION FORM	This material node is only available with the AC/DC Module.	
Compliance matrix	sH (6x6 matrix)	1/Pa

TABLE 2-8: ELECTROMAGNETIC MODELS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Loss factor for compliance matrix sH	eta_sH (6x6 matrix)	1
Piezomagnetic coupling matrix	dHT (3x6 matrix)	m/A
Relative permeability	murT (3x3 matrix)	1
STRESS-MAGNETIZATION FORM	This material node is only available with the AC/DC Module.	
Elasticity matrix	cH (6x6 matrix)	Pa
Loss factor for elasticity matrix cH	eta_cH (6x6 matrix)	1
Piezomagnetic coupling matrix	eHS (3x6 matrix)	T
Relative permeability	murS (3x3 matrix)	1

Equilibrium Discharge

These material property groups for all the material models in the Equilibrium Discharge (including their associated physical properties) can be added to models from the **Material** window. These property groups require the Plasma Module.

TABLE 2-9: EQUILIBRIUM DISCHARGE MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
RADIATION HEAT TRANSFER		
Total volumetric emission coefficient	Qrad	W/m^3

Gas Models

This material property group for an ideal gas (including its associated physical properties) can be added to models from the **Material** page.

TABLE 2-10: GAS MODELS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
IDEAL GAS		
Heat capacity at constant pressure	Cp	J/(kg·K)
Mean molar mass	Mn	kg/mol
Ratio of specific heats	gamma	1
Specific gas constant	Rs	J/(kg·K)

Piezoelectric Models

These material property groups for piezoelectric materials (including their associated physical properties) can be added to models from the **Material** window. These property groups require the Acoustics Module, MEMS Module, or Structural Mechanics Module.

TABLE 2-11: PIEZOELECTRIC MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
STRAIN-CHARGE FORM		
Compliance matrix	sE	1/Pa
Coupling matrix	dET	C/N
Loss factor for compliance matrix	sE	1
Loss factor for coupling matrix	d	1
Loss factor for electrical permittivity	εT	1
Relative permittivity	epsilonrT	1
STRESS-CHARGE FORM		
Coupling matrix	eES	C/m^2
Elasticity matrix	cE	Pa
Loss factor for elasticity matrix	cE	1
Loss factor for coupling matrix	e	1
Loss factor for electrical permittivity	εS	1
Relative permittivity	epsilonrS	1

Piezoresistive Models

These material property groups for piezoresistive materials (including their associated physical properties) can be added to models from the **Material** window. These property groups require the MEMS Module.

TABLE 2-12: GAS MODELS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
ELASTORESISTANCE FORM		
Elastoresistive coupling matrix	ml	Ω·m

TABLE 2-12: GAS MODELS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
PIEZORESISTANCE FORM		
Piezoresistive coupling matrix	PiI	A/m^2

Semiconductors Material Properties

These material property groups for all the material models in semiconductors (including their associated physical properties) can be added to models from the **Material** window. These property groups require the Semiconductor Module.

The Property Group, Variable Names, and SI Unit columns are applicable to all materials in the Semiconductor Module. However, the Values and References columns listed in Table 2-13 are specifically for Silicon in the *COMSOL Multiphysics Reference Manual*.

TABLE 2-13: SEMICONDUCTOR MATERIAL PROPERTIES (ALL MATERIALS) AND VALUES AND REFERENCES FOR SILICON

PROPERTY GROUP AND PROPERTY (ALL MATERIALS)	NAME/VARIABLE (ALL MATERIALS)	SI UNIT	VALUE FOR SILICON	REFERENCE FOR SILICON
BASIC				
Relative permittivity	epsilonr	1	11.7	Ref. 1
Thermal conductivity	k	W/(m·K)	131 W/(m·K)	Ref. 1
Density	rho	kg/m^3	2329 kg/m^3	Ref. 1
Heat capacity at constant pressure	Cp	J/(kg·K)	700 J/(kg·K)	Ref. 1
BAND-GAP NARROWING MODELS>JAIN-ROULSTON MODEL				
Jain-Roulston coefficient (n-type), A	An_jr	V	$3.5 \cdot 10^{-8}$ V	Ref. 12
Jain-Roulston coefficient (n-type), B	Bn_jr	V	0 V	Ref. 12
Jain-Roulston coefficient (n-type), C	Cn_jr	V	0 V	Ref. 12

TABLE 2-13: SEMICONDUCTOR MATERIAL PROPERTIES (ALL MATERIALS) AND VALUES AND REFERENCES FOR SILICON

PROPERTY GROUP AND PROPERTY (ALL MATERIALS)	NAME/VARIABLE (ALL MATERIALS)	SI UNIT	VALUE FOR SILICON	REFERENCE FOR SILICON
Jain-Roulston coefficient (p-type), A	Ap_jr	V	$3.5\cdot10^{-8}$ V	Ref. 12
Jain-Roulston coefficient (p-type), B	Bp_jr	V	0 V	Ref. 12
Jain-Roulston coefficient (p-type), C	Cp_jr	V	0 V	Ref. 12
Band-gap narrowing reference concentration	Nref_jr	$1/m^3$	1 $1/cm^3$	Ref. 12
Conduction band fraction	alpha_jr	1	0.5	Ref. 12
BAND-GAP NARROWING MODELS>SLOTBOOM MODEL				
Band-gap narrowing reference energy	Eref_sb	V	0.00692 V	Ref. 11
Band-gap narrowing reference concentration	Nref_sb	$1/m^3$	$1.3\cdot10^{17}$ $1/cm^3$	Ref. 11
Conduction band fraction	alpha_sb	1	0.5	Ref. 11
GENERATION-RECOMBINATION>AUGER RECOMBINATION				
Auger recombination factor, electrons	Cn	m^6/s	$2.8\cdot10^{-31}$ cm^6/s (valid at 300 K)	Ref. 2
Auger recombination factor, holes	Cp	m^6/s	$9.9\cdot10^{-32}$ cm^6/s (valid at 300 K)	Ref. 2
GENERATION-RECOMBINATION>DIRECT RECOMBINATION				
Direct recombination factor	C	m^3/s	0 m^3/s	N/A

TABLE 2-13: SEMICONDUCTOR MATERIAL PROPERTIES (ALL MATERIALS) AND VALUES AND REFERENCES FOR SILICON

PROPERTY GROUP AND PROPERTY (ALL MATERIALS)	NAME/VARIABLE (ALL MATERIALS)	SI UNIT	VALUE FOR SILICON	REFERENCE FOR SILICON
GENERATION-RECOMBINATION>IMPACT IONIZATION				
a factor, electrons, impact ionization	an	1/V	0.426 1/V	Ref. 3
a factor, holes, impact ionization	ap	1/V	0.243 1/V	Ref. 3
b factor, electrons, impact ionization	bn	V/m	$4.81 \cdot 10^5$ V/cm	Ref. 3
b factor, holes, impact ionization	bp	V/m	$6.53 \cdot 10^5$ V/cm	Ref. 3
c factor, electrons, impact ionization	cn	1/KValues	$3.05 \cdot 10^{-4}$ 1/K	Ref. 3
c factor, holes, impact ionization	cp	1/K	$5.35 \cdot 10^{-4}$ 1/K	Ref. 3
d factor, electrons, impact ionization	dn	1/K	$6.86 \cdot 10^{-4}$ 1/K	Ref. 3
d factor, holes, impact ionization	dp	1/K	$5.67 \cdot 10^{-4}$ 1/K	Ref. 3
GENERATION-RECOMBINATION>SHOCKLEY-READ-HALL RECOMBINATION				
Electron lifetime, SRH	taun	s	10 µs	Ref. 4
Hole lifetime, SRH	taup	s	10 µs	Ref. 4
MOBILITY MODELS>ARORA MOBILITY MODEL				
Electron mobility reference	mun0_ref_arora	$m^2/(V \cdot s)$	1252 $cm^2/(V \cdot s)$	Ref. 5
Hole mobility reference	mup0_ref_arora	$m^2/(V \cdot s)$	407 $cm^2/(V \cdot s)$	Ref. 5
Electron mobility reference minimum	mun_min_ref_arora	$m^2/(V \cdot s)$	88 $cm^2/(V \cdot s)$	Ref. 5
Hole mobility reference minimum	mup_min_ref_arora	$m^2/(V \cdot s)$	53.4 $cm^2/(V \cdot s)$	Ref. 5
Electron reference impurity concentration	Nn0_ref_arora	$1/m^3$	$1.26 \cdot 10^{17}$ $1/cm^3$	Ref. 5

TABLE 2-13: SEMICONDUCTOR MATERIAL PROPERTIES (ALL MATERIALS) AND VALUES AND REFERENCES FOR SILICON

PROPERTY GROUP AND PROPERTY (ALL MATERIALS)	NAME/VARIABLE (ALL MATERIALS)	SI UNIT	VALUE FOR SILICON	REFERENCE FOR SILICON
Hole reference impurity concentration	Np0_ref_arora	$1/m^3$	$2.35 \cdot 10^{17}$ $1/cm^3$	Ref. 5
Alpha coefficient	alpha0_arora	1	0.88	Ref. 5
Mobility reference minimum exponent	beta1_arora	1	-0.57	Ref. 5
Mobility reference exponent	beta2_arora	1	-2.33	Ref. 5
Impurity concentration reference exponent	beta3_arora	1	2.4	Ref. 5
Alpha coefficient exponent	beta4_arora	$m^2/(V \cdot s)$	-0.146	Ref. 5
Reference temperature	Tref_arora	K	300 K	Ref. 5
MOBILITY MODELS>CAUGHEY-THOMAS MOBILITY MODEL				
Electron alpha coefficient	alphan0_ct	1	1.11	Ref. 6
Electron alpha exponent	betan1_ct	1	0.66	Ref. 6
Electron saturation velocity	vn0_ct	m/s	$1 \cdot 10^7$ cm/s	Ref. 6
Electron velocity saturation exponent	betan2_ct	1	-0.87	Ref. 6
Hole alpha coefficient	alphap0_ct	1	1.21	Ref. 6
Hole alpha exponent	betap1_ct		0.17	Ref. 6
Hole saturation velocity	vp0_ct	m/s	$8.37 \cdot 10^6$ cm/s	Ref. 6
Hole velocity saturation exponent	betap2_ct	1	-0.52	Ref. 6
Reference temperature	Tref_ct	K	300 K	Ref. 6

TABLE 2-13: SEMICONDUCTOR MATERIAL PROPERTIES (ALL MATERIALS) AND VALUES AND REFERENCES FOR SILICON

PROPERTY GROUP AND PROPERTY (ALL MATERIALS)	NAME/VARIABLE (ALL MATERIALS)	SI UNIT	VALUE FOR SILICON	REFERENCE FOR SILICON
MOBILITY MODELS>FLETCHER MOBILITY MODEL				
Fletcher mobility coefficient 1	F1_fl	1/(cm·V·s)	1.04×10^{21} 1/(cm·V·s)	Ref. 7
Fletcher mobility coefficient 2	F2_fl	$1/m^2$	7.45×10^{13} $1/cm^2$	Ref. 7
Reference temperature	Tref_fl	K	300 K	Ref. 7
MOBILITY MODELS>LOMBARDI SURFACE MOBILITY MODEL				
Electron delta coefficient	deltan_ls	V/s	5.82×10^{14} V/s	Ref. 8
Electron mobility reference	mun1_ls	$m^2/(V·s)$	$4.75 \times 10^7 cm^2/(V·s)$	Ref. 8
Electron mobility reference	mun2_ls	$m^2/(V·s)$	1.74×10^5 $cm^2/(V·s)$	Ref. 8
Electron alpha coefficient	alphan_ls	1	0.125	Ref. 8
Hole delta coefficient	deltap_ls	V/s	2.05×10^{14} V/s	Ref. 8
Hole mobility reference	mup1_ls	$m^2/(V·s)$	9.93×10^7 $cm^2/(V·s)$	Ref. 8
Hole mobility reference	mup2_ls	$m^2/(V·s)$	8.84×10^5 $cm^2/(V·s)$	Ref. 8
Hole alpha coefficient	alphap_ls	1	0.0317	Ref. 8
Reference temperature	Tref_ls	K	1 K	Ref. 8
Electric field reference	Eref_ls	V/m	1 V/cm	Ref. 8
Doping concentration reference	Nref_ls	$1/m^3$	1 $1/cm^3$	Ref. 8
MOBILITY MODELS>POWER LAW MOBILITY MODEL				
Electron mobility reference	mun0_pl	$m^2/(V·s)$	1448 $cm^2/(V·s)$	Ref. 5

TABLE 2-13: SEMICONDUCTOR MATERIAL PROPERTIES (ALL MATERIALS) AND VALUES AND REFERENCES FOR SILICON

PROPERTY GROUP AND PROPERTY (ALL MATERIALS)	NAME/VARIABLE (ALL MATERIALS)	SI UNIT	VALUE FOR SILICON	REFERENCE FOR SILICON
Hole mobility reference	mup0_pl	$m^2/(V \cdot s)$	473 $cm^2/(V \cdot s)$	Ref. 5
Electron exponent	alphan_pl	1	2.33	Ref. 5
Hole exponent	alphap_pl	1	2.23	Ref. 5
Reference temperature	Tref_pl	K	300 K	Ref. 5
SEMICONDUCTOR MATERIAL				
Band gap	Eg0	V	1.12 V (valid at 300 K)	Ref. 1
Effective density of states, conduction band	Nc	$1/m^3$	2.8×10^{19} $1/cm^3$ $\times(T/300\ K)^{3/2}$	Ref. 1
Effective density of states, valence band	Nv	$1/m^3$	1.04×10^{19} $1/cm^3$ $\times(T/300\ K)^{3/2}$	Ref. 1
Electron affinity	chi0	V	4.05 V	Ref. 1
Electron mobility	mun	$m^2/(V \cdot s)$	1450 $cm^2/(V \cdot s)$	Ref. 1
Hole mobility	mup	$m^2/(V \cdot s)$	500 $cm^2/(V \cdot s)$	Ref. 1

Solid Mechanics Material Properties

These material property groups for material models in solid mechanics (including their associated physical properties) can be added to models from the **Material** window. Most of these properties require the Structural Mechanics Module. The property groups of the external material are of a special type that depends on the selected interface type and are not individually documented.

TABLE 2-14: SOLID MECHANICS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
LINEAR ELASTIC MATERIAL		
ANISOTROPIC		
Elasticity matrix	D	Pa
Loss factor for elasticity matrix D	eta_D	1

TABLE 2-14: SOLID MECHANICS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
ANISOTROPIC, VOIGT NOTATION		
Elasticity matrix, Voigt notation	DV0	Pa
Loss factor for elasticity matrix D, Voigt notation	eta_DVo	1
BULK MODULUS AND SHEAR MODULUS		
Bulk modulus	K	N/m^2
Shear modulus	G	N/m^2
LAMÉ PARAMETERS		
Lamé parameter λ	lambLame	N/m^2
Lamé parameter μ	muLame	N/m^2
ORTHOTROPIC		
Young's modulus	Evector	Pa
Poisson's ratio	nuvector	1
Shear modulus	Gvector	N/m^2
Loss factor for orthotropic Young's modulus	eta_Evector	1
Loss factor for orthotropic shear modulus	eta_Gvector	1
ORTHOTROPIC, VOIGT NOTATION		
Shear modulus, Voigt notation	GvectorVo	N/m^2
Loss factor for orthotropic shear modulus, Voigt notation	eta_GvectorVo	1
PRESSURE-WAVE AND SHEAR-WAVE SPEEDS		
Pressure-wave speed	cp	m/s
Shear-wave speed	cs	m/s
YOUNG'S MODULUS AND POISSON'S RATIO		
Young's modulus	E	Pa
Poisson's ratio	nu	1
YOUNG'S MODULUS AND SHEAR MODULUS		
Young's modulus	E	Pa
Shear modulus	G	N/m^2
LINEAR VISCOELASTIC MATERIAL		
Long-term shear modulus	Gv	N/m^2
Bulk modulus	K	N/m^2
POROELASTIC MATERIAL		
Biot-Willis coefficient	alphaB	1
Porosity	epsilon	1

TABLE 2-14: SOLID MECHANICS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Permeability	kappa	m^2
SAFETY		
ISOTROPIC STRENGTH PARAMETERS		
Tensile strength	sigmat	Pa
Compressive strength	sigmac	Pa
Biaxial compressive strength	sigmabc	Pa
ISOTROPIC ULTIMATE STRAINS		
Ultimate tensile strain	epsilont	1
Ultimate compressive strain	epsilonc	1
ORTHOTROPIC STRENGTH PARAMETERS, VOIGT NOTATION		
Tensile strengths	sigmats	Pa
Compressive strengths	sigmacs	Pa
Shear strengths	sigmass	Pa
ORTHOTROPIC ULTIMATE STRAINS, VOIGT NOTATION		
Ultimate tensile strains	epsilonts	1
Ultimate compressive strains	epsiloncs	1
Ultimate shear strains	gammass	1
ANISOTROPIC STRENGTH PARAMETERS, VOIGT NOTATION		
Second rank tensor, Voigt notation	F_s	1/Pa
Fourth rank tensor, Voigt notation	F_f	$m^2 \cdot s^4/kg^2$

- The *Structural Mechanics Module User's Guide* and Table 2-17
- The *Structural Mechanics Module User's Guide* and Table 2-15
- The *Fatigue Module User's Guide* and Table 2-16

Solid Mechanics Material Properties: Nonlinear Structural Materials Module

These material property groups for material models in solid mechanics using the Nonlinear Structural Materials Module (including their associated physical properties) can be added to models from the **Material** window.

TABLE 2-15: HYPERELASTIC AND ELASTOPLASTIC MATERIAL PROPERTIES

PROPERTY GROUP AND PROPERTY	**NAME/VARIABLE**	**SI UNIT**
ELASTOPLASTIC MATERIAL		
Hardening function	sigmagh	Pa
Hill's coefficients	Hillcoefficients	$(m^2 \cdot s^4)/kg^2$
Initial tensile and shear yield stresses	ys	N/m^2
Initial yield stress	sigmags	Pa
Isotropic tangent modulus	Et	Pa
Kinematic tangent modulus	Ek	Pa
ARMSTRONG-FREDERICK		
Kinematic hardening modulus	Ck	Pa
Kinematic hardening parameter	gammak	1
CHABOCHE		
Kinematic hardening modulus	Ck0_cha	Pa
LUDWIK		
Strength coefficient	k_lud	Pa
Hardening exponent	n_lud	1
SWIFT		
Reference strain	e0_swi	1
Hardening exponent	n_swi	1
VOCE		
Saturation flow stress	sigma_voc	Pa
Saturation exponent	beta_voc	1
HOCKETT-SHERBY		
Steady-state flow stress	sigma_hoc	Pa
Saturation coefficient	m_hoc	1
Saturation exponent	n_hoc	1
CREEP		
NORTON		

TABLE 2-15: HYPERELASTIC AND ELASTOPLASTIC MATERIAL PROPERTIES

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Creep rate coefficient	A_nor	1/s
Reference stress	sigRef_nor	Pa
Stress exponent	n_nor	1
GAROFALO (HYPERBOLIC SINE)		
Creep rate coefficient	A_gar	1/s
Reference stress	sigRef_gar	Pa
Stress exponent	n_gar	1
NABARRO-HERRING		
Volume diffusivity	D_nav	m^2/s
Burgers vector	b_nav	m
Grain diameter	dg_nav	m
COBLE		
Ionic diffusivity	D_cob	m^2/s
Burgers vector	b_cob	m
Grain diameter	dg_cob	m
WEERTMAN		
Diffusivity	D_wee	m^2/s
Burgers vector	b_wee	m
Stress exponent	n_wee	1
Reference stress	sigRef_wee	Pa
VISCOPLASTIC MATERIAL		
ANAND		
Viscoplastic rate coefficient	A_ana	1/s
Activation energy	Q_ana	J/mol
Multiplier of stress	xi_ana	1
Stress sensitivity	m_ana	1
Deformation resistance saturation coefficient	s0_ana	Pa
Deformation resistance initial value	sa_init	Pa
Hardening constant	h0_ana	Pa
Hardening sensitivity	a_ana	1
Deformation resistance sensitivity	n_ana	1
CHABOCHE		
Viscoplastic rate coefficient	A_cha	1/s

TABLE 2-15: HYPERELASTIC AND ELASTOPLASTIC MATERIAL PROPERTIES

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Reference stress	sigRef_cha	Pa
Stress exponent	n_cha	1
PREZYNA		
Viscoplastic rate coefficient	A_per	1/s
Reference stress	sigRef_per	Pa
POROPLASTIC MATERIAL		
Initial yield stress	sigmags	Pa
Shima-Oyane alpha parameter	alphaShima	1
Shima-Oyane gamma parameter	gammaShima	1
Shima-Oyane m parameter	mShima	1
Initial void volume fraction	f0	1
Critical void volume fraction	fc	1
Failure void volume fraction	ff	1
Tvergaard correction coefficient q1	q1GTN	1
Tvergaard correction coefficient q2	q2GTN	1
Tvergaard correction coefficient q3	q3GTN	1
Maximum void volume fraction	fmax	1
NONLINEAR ELASTIC MATERIAL		
Reference stress	sigRef	Pa
Reference strain	eRef	1
Stress exponent	n_stress	1
Reference shear strain	gammaRef	1
Strain exponent	n_strain	1
Bulk modulus in tension	Kt	Pa
Bulk modulus in compression	Kc	Pa
Ultimate deviatoric stress	q_ult	Pa
Ultimate strain	e_ult	1
ELASTOPLASTIC SOIL MATERIAL		
CAM-CLAY		
Swelling index	kappaSwelling	1
Compression index	lambdaComp	1
Void ratio at reference pressure	evoidref	1

TABLE 2-15: HYPERELASTIC AND ELASTOPLASTIC MATERIAL PROPERTIES

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Slope of critical state line	M	1
STRUCTURED CAM-CLAY		
Swelling index for structured clay	kappaSwellingS	1
Compression index for destructured clay	lambdaCompS	1
Void ratio at reference pressure for destructured clay	evoidrefS	1
Destructuring index for volumetric deformation	dvS	1
Destructuring index for shear deformation	dsS	1
Slope of critical state line	M	1
Additional void ratio at initial yielding	Deltaei	1
Initial structure strength	pbi	Pa
Plastic potential shape parameter	zetaS	1
Critical effective deviatoric plastic strain	epdevc	1
BARCELONA BASIC		
Swelling index	kappaSwelling	1
Swelling index for changes in suction	kappaSwellings	1
Compression index at saturation	lambdaComp0	1
Weight parameter	wB	1
Soil stiffness parameter	mB	Pa
Plastic potential smoothing parameter	bB	1
Tension to suction ratio	kB	1
Void ratio at reference pressure and saturation	evoidref0	1
Initial yield value for suction	sy0	Pa
HARDENING SOIL		
Reference stiffness for primary loading	E50Ref	Pa
Reference stiffness for unloading and reloading	EurRef	Pa
Stress exponent	mH	1
Bulk modulus in compression	Kc	Pa
Void ratio at reference pressure	evoidref	1
HYPERELASTIC MATERIALS		
ARRUDA-BOYCE		
Macroscopic shear modulus	mu0	N/m^2

TABLE 2-15: HYPERELASTIC AND ELASTOPLASTIC MATERIAL PROPERTIES

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Number of segments	Nseg	1
BLATZ-KO		
Model parameters	phiBK	1
Model parameters	betaBK	1
Shear modulus	muBK	Pa
GAO		
Model parameters	aG	Pa
Model parameters	nG	1
GENT		
Macroscopic shear modulus	muG	Pa
Model parameters	jmG	1
MOONEY-RIVLIN		
Model parameters	C01, C02, C03, C10, C11, C12, C20, C21, C30	Pa
MURNAGHAN	The Murnaghan node adds five model parameters. The model is based on strain invariants and is typically used in acoustoelasticity.	
Murnaghan third-order elastic moduli	l	Pa
Murnaghan third-order elastic moduli	m	Pa
Murnaghan third-order elastic moduli	n	Pa
Lamé parameter λ	lambLame	Pa
Lamé parameter μ	muLame	Pa
VARGA		
Model parameters	c1VA	Pa
Model parameters	c2VA	Pa
YEOH		
Model parameters	c1YE	Pa
Model parameters	c2YE	Pa
Model parameters	c3YE	Pa

Solid Mechanics Material Properties: Fatigue Module

These material property groups for material models in solid mechanics using the Fatigue Module (including their associated physical properties) can be added to models from the **Material** window.

TABLE 2-16: ELASTOPLASTIC AND FATIGUE BEHAVIOR MATERIAL PROPERTIES

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
ELASTOPLASTIC MATERIAL>RAMBERG-OSGOOD		
Cyclic hardening coefficient	K_ROcyclic	Pa
Cyclic hardening coefficient	n_ROcyclic	1
FATIGUE BEHAVIOR>ENERGY-BASED		
DARVEAUX		
Crack initiation energy coefficient	K1_Darveaux	1
Crack initiation energy exponent	k2_Darveaux	1
Crack propagation energy coefficient	K3_Darveaux	m
Crack propagation energy exponent	k4_Darveaux	1
Reference energy density	Wref_Darveaux	J/m^3
MORROW		
Fatigue energy coefficient	Wf_Morrow	J/m^3
Fatigue energy exponent	m_Morrow	1
FATIGUE BEHAVIOR>FATIGUE BEHAVIOR>APPROXIMATE S-N CURVE		
Transition stress	sigmat	Pa
Transition life	Nt	1
Endurance life	Ne	1
FATIGUE BEHAVIOR>GENERAL		
Endurance limit	sigmae	Pa
FATIGUE BEHAVIOR>STRAIN-BASED		
COFFIN-MANSON		
Fatigue ductility coefficient	epsilonf_CM	1
Fatigue ductility exponent	c_CM	1
Shear fatigue ductility coefficient	gammaf_CM	1
Shear fatigue ductility exponent	cgamma_CM	1
FATEMI-SOCIE		

TABLE 2-16: ELASTOPLASTIC AND FATIGUE BEHAVIOR MATERIAL PROPERTIES

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Normal stress sensitivity coefficient	k_FS	1
WANG-BROWN		
Normal stress sensitivity coefficient	S_WB	1
FATIGUE BEHAVIOR>STRESS-BASED		
BASQUIN		
Fatigue strength coefficient	sigmaf_Basquin	Pa
Fatigue strength exponent	b_Basquin	1
Shear fatigue strength coefficient	tauf_Basquin	Pa
Shear fatigue strength exponent	bgamma_Basquin	1
FINDLEY		
Normal stress sensitivity coefficient	k_Findley	1
Limit factor	f_Findley	Pa
MATAKE		
Normal stress sensitivity coefficient	k_Matake	1
Limit factor	f_Matake	Pa
NORMAL STRESS		
Limit factor	f_NormalStress	Pa
DANG VAN		
Hydrostatic stress sensitivity coefficient	a_DangVan	1
Limit factor	b_DangVan	Pa

Solid Mechanics Material Properties: Geomechanics Material Model

These material property groups for material models in solid mechanics (including their associated physical properties) can be added to models from the **Material** window. These property groups require the Geomechanics Module.

TABLE 2-17: GEOMECHANICS MODELS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
DRUCKER-PRAGER		
Drucker-Prager alpha coefficient	alphaDrucker	1

TABLE 2-17: GEOMECHANICS MODELS MATERIALS

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Drucker-Prager k coefficient	kDrucker	Pa
HOEK BROWN		
Hoek-Brown m parameter	mHB	1
Hoek-Brown s parameter	sHB	1
Geological strength index	GSI	1
Disturbance factor	Dfactor	1
Intact rock parameter	miHB	1
LADE-DUNCAN		
Lade-Duncan k coefficient	kLade	1
MATSUOKA-NAKAI		
Matsuoka-Nakai mu coefficient	muMatsuoka	1
MOHR-COULOMB		
Cohesion	cohesion	Pa
Angle of internal friction	internalphi	rad
OTTOSEN		
Ottosen a parameter	aOttosen	1
Ottosen b parameter	bOttosen	1
Size factor	k1Ottosen	1
Shape factor	k2Ottosen	1
YIELD STRESS PARAMETERS		
Uniaxial tensile strength	sigmaut	Pa
Uniaxial compressive strength	sigmauc	Pa
Biaxial compressive strength	sigmabc	Pa

Thermal Expansion Material Properties

This material property group for thermal expansion properties can be added to models from the **Material** page.

TABLE 2-18: THERMAL EXPANSION MATERIAL PROPERTIES

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
IDEAL GAS		
Isotropic tangent coefficient of thermal expansion	alphatanIso	1/K
Isotropic thermal strain	dLIso	1

TABLE 2-18: THERMAL EXPANSION MATERIAL PROPERTIES

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Tangent coefficient of thermal expansion	alphatan_iso; alphatanij	1/K
Thermal strain	dLi_iso, dLij	1

External Material Properties

The property groups of the external materials are of a special type that depends on the selected interface type and are not individually documented. You can incorporate as many parameters in the call to the external DLL when you add an external material, these parameters will appear in the Material node as material inputs, see Working with External Materials for more information.

Using Functions

The Material Library describes material properties with functions, usually functions of temperature, and for this purpose it uses piecewise analytic functions (polynomials). For user-defined property functions, three types of functions can be defined: analytic functions, piecewise analytic functions, and interpolation functions.

Functions are useful for describing material properties as, for example, functions of temperature or pressure.

Adding a Function to the Material

Material functions are either automatically added to the **Model Builder** sequence (usually with materials from the material library) or functions can be added based on individual requirements.

1 Add a material to the **Component** node (see The Material Browser Window and The Add Material Window).
2 Add an **Analytic** ( ), **Interpolation** ( ), or **Piecewise** ( ) function.

Win

To add an **Analytic** ( ), **Interpolation** ( ), or **Piecewise** ( ) function:

- On the **Materials** toolbar, click **Analytic**, **Interpolation**, or **Piecewise**.
- Right-click a property group node (for example, **Basic**) and select a function from the **Functions** list.

Mac

Linux

To add an **Analytic** ( ), **Interpolation** ( ), or **Piecewise** ( ) function:

- Right-click a property group node, for example, **Basic** and select a function from the **Functions** list.
- On the **Materials** contextual toolbar, click **Analytic**, **Interpolation**, or **Piecewise**.

- Select **Analytic** to add an analytic function of one or more input arguments.
- Select **Interpolation** to add an interpolation function that can interpolate from structured data (defined on a grid) or unstructured data (defined on a generic point cloud).
- Select **Piecewise** to add a piecewise function that is useful if a material property has different definitions on different intervals. The intervals must not overlap, and there cannot be any holes between intervals.

- Defining an Analytic Function
- Analytic, Interpolation, and Piecewise in the *COMSOL Multiphysics Reference Manual*

Once a function is created, you can use it for any property in the same property group.

Defining an Analytic Function

Assume that you want to define the density ρ_1 for a material as a function of pressure and temperature: $\rho_1 = \rho_1(p,T)$. You can name the function `rho1(p,T)` and use the expression `p*0.02897/8.314/T` to define the function.

1 On the **Materials** toolbar, click the **Browse Materials** , **Add Material** , or **Blank Material** button to add a new material to the Component (or use an existing material where density is not defined, or redefine the current expression for the density).

2 Add a **Density** property to the material.

  a In the **Model Builder**, click the **Material** node.

  b In the **Settings** window for **Material**, click to expand the **Material Properties** section. Under **Basic Properties**, right-click **Density** and **Add to Material**.

  A **Density** property is added to the **Basic** property group.

3 In the **Model Builder**, under the material node, right-click **Basic** and select **Functions>Analytic**. This adds an **Analytic** subnode ( ) under **Basic**.

4 On the **Settings** window for **Analytic**, enter `rho1` in **Function name**. Replace the default name.

5 Under the **Definition** section:

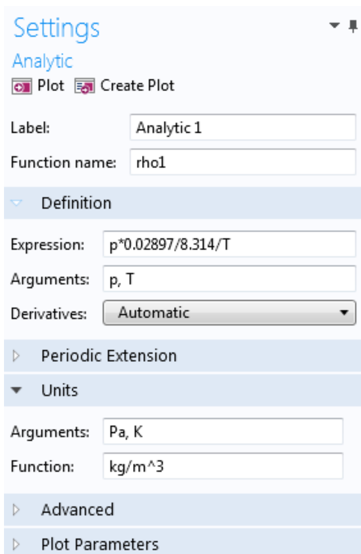
a In the **Expression** field, enter `p*0.02897/8.314/T`.

b In the **Arguments** column, enter `p,T`.

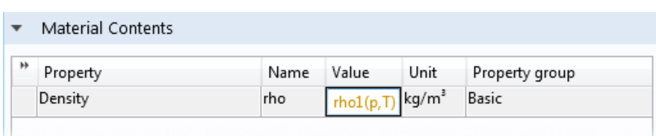
6 Under **Units**:

a In the **Arguments** field, enter `Pa, K` as the units for the pressure and the temperature, respectively.

b In the **Function** field, enter `kg/m^3` as the unit for the function's output (density). The function `rho1` can now be used to define the density in your material.



7 Click the **Material** node. In the **Settings** window for **Material**, under **Material Contents**, enter `rho1(p,T)` in the **Value** column (in the **Density** row).



Click the **Basic** node to notice that the **Density** analytic function is defined in the **Settings** window for **Property Group** under **Output Properties**. The expression will be orange if there are no variables `p` and `T` for pressure and temperature, respectively, defined in the component. See Figure 2-12.

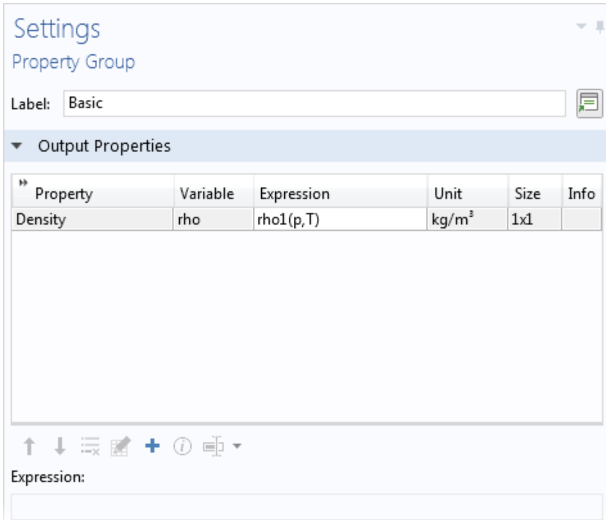


Figure 2-12: A density property is defined using an analytic function.

Index