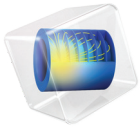


Material Library

User's Guide

6.4

COMSOL

Material Library User's Guide

Version: COMSOL 6.4

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 181,000 material property function datasets; these datasets specify various material properties of more than 18,800 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 online resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or context-based) help, and the Application Libraries are all accessed through the COMSOL Desktop.

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

THE DOCUMENTATION AND ONLINE HELP

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

Opening Topic-Based Help

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

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

Win

To open the **Help** window:

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

Mac

Linux

To open the **Help** window:

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

Opening the Documentation Window

Mac

Linux

To open the **Documentation** window:

- Press Ctrl+F1.
- In the main toolbar, click the **Documentation** ( ) button.
- From the main menu, select **Help** > **Documentation**.

THE APPLICATION LIBRARIES WINDOW

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

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

Opening the Application Libraries Window

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

CONTACTING COMSOL BY EMAIL

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

COMSOL ACCESS AND TECHNICAL SUPPORT

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and a case number will be sent to you by email. You can also access technical support, software updates, license information, and other resources by registering for a COMSOL Access account.

COMSOL ONLINE RESOURCES

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

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.

In this chapter:

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, piezoresistive, and semiconductor 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** :

- In 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** :

- In 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.

Right-click a material library in the **Material Browser** and choose **Reload Selected** ( ) to clear all cached data for that library and force the COMSOL Multiphysics software to reload the content from the file system. This operation is useful, for example, if the library is a user-defined library that has been edited since the COMSOL Multiphysics session started and needs to be reloaded to display the latest contents.

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, primarily for the Material Library product, by UNS, DIN, WNr, JIS, ASTM, AISI, SAE, AMS, or AFNOR 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, DIN, or other material identification number, composition, and orientation or other variations.

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 and any other materials where property information has been added, reference information or other 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. 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. You can also click the downward-pointing arrow to the right of the Import Material Library button and choose **Import Material From** ( ) to open the fullscreen **Select Material** window and choose the material from a database. Go to Importing a Material Library in the *COMSOL Multiphysics Reference Manual*.

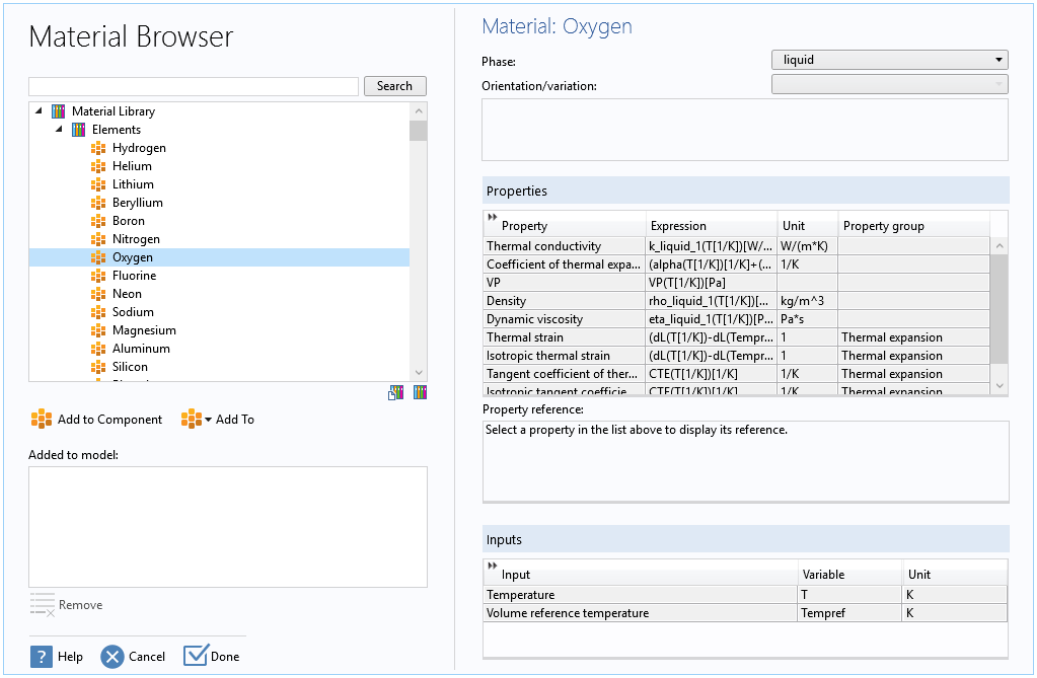


Figure 2-1: The Material Browser details a material's properties after selection. In this example, the properties of Oxygen are listed to the right of the Material Browser folders.

MATERIAL LIBRARY FOLDERS

TABLE 2-1: MATERIAL LIBRARY FOLDERS.

FOLDER
Elements
Iron Alloys
Nickel Alloys
Copper Alloys
Controlled Expansion Alloys
Aluminum Alloys
Magnesium Alloys
Titanium Alloys
Oxides
Complex Oxides, Silicates, and Slags
Carbides
Fluorides
Hydrides
Cermets
Tool Steels
Carbons
Thermal Insulation Materials
Intermetallics
Refractory Metal Alloys
High Entropy and Equiatomic Alloys
Thermal Coatings
Nylons and PA/PI (polyamides/polyimides)
PAI (polyamide-imide)
PPA (polyphthalamides)
Polyesters
PEI (polyetherimide)
PARA (polyarylamide)
PLA (polylactic acid)
LCP (liquid crystal polymer)
Acetal/POM (polyoxymethylene)

TABLE 2-1: MATERIAL LIBRARY FOLDERS.

FOLDER
PVF/PVDF/PVC (poly(vinyl/idene fluoride/chloride))
EVA (ethylene-vinyl acetate)
PC (polycarbonates)
UR, PU, and TPU (thermoset polyurethanes)
Miscellaneous Polymers
Parylenes and Polyolefins
Miscellaneous Polymer Composites
Circuit Boards and Antenna Substrates
Silicones and Elastomers
DAP and DAIP (diallyl (ortho-, iso-) phthalates
Epoxies, Adhesives, and Underfills
PMMA, PMMI, and Acrylics
Minerals, Rocks, and Soils
Woods
PP (polypropylenes)
PET (polyethylene terephthalate)
PCT (polycyclohexylenedimethylene terephthalate)
TPC (thermoplastic copolyester elastomer)
Structural Foams
PBT (polybutylene terephthalate)
ECTFE (polyethylene chlorotrifluoroethylene)
PFA and ETFE (fluorocarbons)
PE (polyethylene)
M-PPE (modified polyphenylene ether)
PPE/PA (polyphenylene ether/polyamide)
PPE/PP (polyphenylene ether/polypropylene)
PPE/TPE, TPV, and SEBS
PPE/PS (polyphenylene ether/polystyrene)
ABS (acrylonitrile butadiene styrene)
TCP (thermally conductive polymers)
PEBA (polyether block amide)

TABLE 2-1: MATERIAL LIBRARY FOLDERS.

FOLDER
PAEK and PEEK (polyaryletherketone)
PSU (polysulfone)
PES/PESU (polyethersulfone)
PPS (polyphenylene sulfide)
PPSU/PPSF (polyphenysulfone)
Zinc Alloys
Solders, Brazes, and Low Melting Alloys
Precious Metal and Rare Earth Alloys
Thermocouple Alloys
Semi/Superconductors and Optical Materials
Organics, Hydrocarbons, and Lubricants
Organic and Hydrocarbon Mixtures
Foods
Other Materials
Cobalt Alloys
Resistance Alloys
Bimetal Alloys
Magnetic Alloys and Electrical Steels
Metal Matrix Composites
Ceramic Matrix Composites
Salts (polyatomic)
Salts (monatomic)
Salts (binary mixtures)
Salts (ternary mixtures)
Electroceramics
Thermoelectrics
Silicides
Borides
Glasses
Metallic Glasses
Nitrides and Oxynitrides

TABLE 2-1: MATERIAL LIBRARY FOLDERS.

FOLDER
Cast Irons
Acids
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 material numbers such as a UNS number or DIN number, for materials that have such material numbers defined.

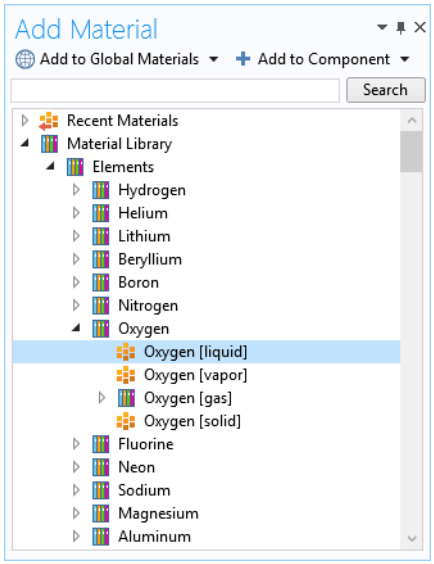


Figure 2-2: The Add Material window. In this example, the liquid phase of Oxygen is selected and can be added to the Material node in the local Component or as a global material in the Model Builder.

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 or material 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 included for some materials in the Material Library product, for example). 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. Choose Link in Global Materials, for example, to create a material

library link instead of a material. 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.

Merging a Material Into Another Material

You can merge a material into another material. The other material, into which that material is merged, then inherits all material properties from the material that you merge. If both materials include the same material properties, then the values of those material properties in the material that you merge into another material are used in the other, remaining material. The **Material** node for the node that you merge into another material is removed from the model tree. To merge a material, do the following steps:

1 Right-click the **Material** node for the material that you want to merge into another material in the model.

2 From the **Merge Into** context menu, choose one of the materials in the same folder (under **Global Materials**, or under **Materials** in a **Component**), or choose a material from another folder.

3 The material is then merged into the material that you selected from the **Merge Into** menu. The material property values from this **Material** node is then used in the other material, and the original material property values are overwritten if both materials included some common material properties. This **Material** node is removed.

You can also use the **Merge Into** menu ( ) in the **Materials** toolbar.

Importing and Exporting Materials

You can import and export materials to and from your COMSOL Multiphysics models. The **Materials** toolbar include **Import Materials** and **Export Materials** buttons. You can also right-click one of the following nodes in the Model Builder and choose **Import Materials**, **Import Materials From**, or **Export Materials**:

- The **Materials** node under **Global Definitions**.

- The **Materials** node under **Definitions** in a component.
- Any node group under those branches.

The selected node determines where the imported materials will go or what materials to export into a new XML-file.

To import materials first select one of these nodes, then click the **Import Materials** button or choose it from a context menu. An **Import Materials** dialog appears that displays the content of the selected file. Select the materials to import and then click **OK** to import them to the model. You can also choose **Import Materials From**, which opens the **Select Model** fullscreen window for choosing a materials file from a database.

Some types of materials can only be imported to either **Global Definitions** or **Definitions**. Importing such a materials to the wrong branch is not allowed, so they are therefore ignored in the import. A message will appear when this happens.

To export materials, click the **Export Materials** button or choose it from a context menu. In the **Export Materials** dialog that appears, choose the file location and enter a name for the XML-file to export materials to. Click **OK** when you are done to export the materials in the chosen file.

MATERIAL IMPORT AND EXPORT FORMATS

COMSOL can import materials from the following formats:

- MPH files.
- Model XML-file format.
- A special XML-file format (JmatPro XML) for metal processing.

There are no special extension for the supported XML-file formats, and they expect the `.xml` extension. The COMSOL Multiphysics software detects the actual format automatically by looking at the first part of the file.

For materials export, COMSOL Multiphysics only supports explicit export to the Model XML-file format. You can export materials on the MPH-file format by just saving a model, which can contain materials only.

For more information about the XML-file format, see Materials in *The COMSOL File Formats* chapter in the *COMSOL Multiphysics Programming Reference Manual*.

To see how a material is stored in the Model XML-file format, you can export it from the COMSOL Desktop and then open the resulting XML-file to see the XML content for the exported materials.

Materials

Use the nodes under **Materials** ( ) to add predefined or user-defined materials, including layered materials and porous 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, About the Material Libraries, Layered Material, Layered Material Link, Layered Material Stack, Single Layer Materials, Porous Material, and Topology Link in the *COMSOL Multiphysics Reference Manual*.

You can right-click the **Materials** node and select **Add Material 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. You 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.

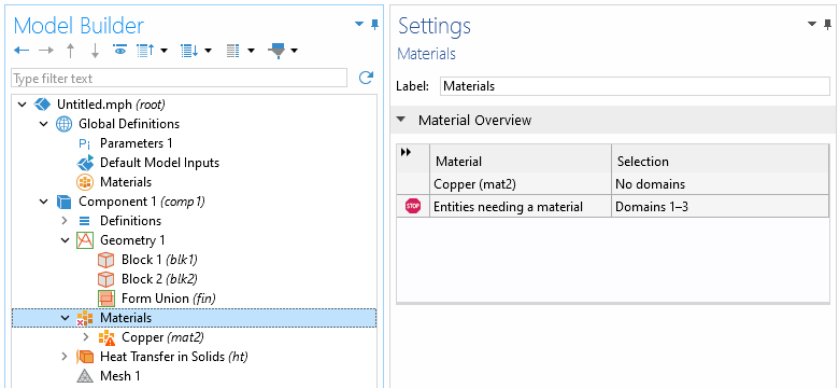


Figure 2-3: An example of a Materials node error.

Grouping of Materials for User-Defined Material Libraries

When creating materials for a user-defined material library, you can add **Group** nodes to create folders in the material library or to define a phase or a phase and orientation in the **Material Library Settings** section.

This structure will appear in the **Material Browser** window as folders and materials with a selection of phase or orientation/variation when you have saved the materials with their settings and group structure as an MPH file and then imported that file as a user-defined material library.

When you add grouping for the phase and for an orientation or variation, use the **Label** in the **Group** node for the name of the material, such as `Material name`. For any subgroups or **Material** nodes for the phase, use `Material name [Phase]` as the **Label** for those nodes. For **Material** nodes under those phase groups, use `Material name [Phase, Orientation/variation]` as the **Label** to make the material appear with those phase and orientation or variation names (in brackets) as options in the **Phase** and **Orientation/variation** lists in the **Material Browser**.

Select the **Material Library Settings** checkbox in the **Show More Options** dialog to display this section for material under **Global Definitions**.

From the **Group type** list, choose one of the following options:

- **Library folder** (the default), to use the **Group** node's label as a library folder for organizing the material library into groups of material.
- **Phase**, to use the group for defining variations of a material with different material phases and possibly also orientation or variation. You define those properties in the **Material** subnodes' **Material Library Settings** sections.
- **Phase and orientation**, to use the group for materials with a certain phase and different orientation or variation. You define the orientation or variation in the **Material** subnodes' **Material Library Settings** sections. From the **Phase** list, choose **Custom**, **Solid**, **Liquid**, or **Gas** to define the phase of the materials in this group.

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 Library Link, 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.

A standard **Material** node in the global component can turn into a layered material by adding a **Shell** property group. After that, it can be linked by a Layered Material Link

In the **Label** field, enter a suitable label for the material. The label also provides the name that will be shown when the material is added to a user-defined material library. See also **Material Library Settings** below. You can also change the **Name** of the material.

This changes both the tag of the material feature and the namespace in which it will define its variables and functions.

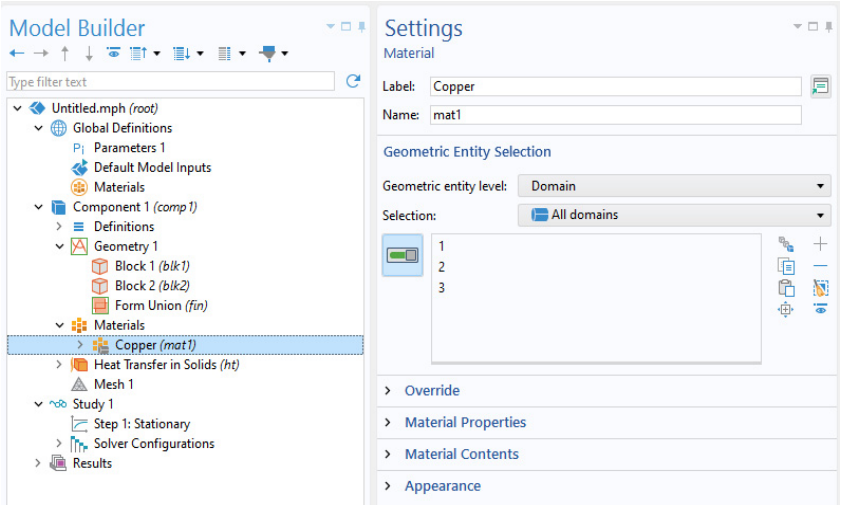


Figure 2-4: Click the Copper node to open the Settings window for Material for the node.

GEOMETRIC ENTITY SELECTION

This section is available for materials in a component. 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). The icon for the Material node indicates the geometry level. For a 3D component, for example, the following icon overlays represent the domain, boundary, edge, and point level, respectively: , , , and .

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, available for materials in a component, 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

ORIENTATION AND POSITION

This section only appears in **Material** nodes that are single layer material. See Single Layer Materials.

Select a **Coordinate system** defining the principal directions of the laminate. Only **Boundary System** coordinate systems can be selected.

Choose a **Position** — **Midplane on boundary**, **Downside on boundary**, **Upside 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 **Downside on boundary**, and the value –1 corresponds to **Upside 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 2-10, but there is only a single layer. You can also click the downward pointing arrow to choose **Layer Cross Section Preview** ( ) button or **Create Layer Cross Section Preview** ( ) button, which adds the preview plot as a new plot group under **Results**.

MATERIAL PROPERTIES

Each material node defines a number of output properties. The property definitions, including expressions, functions and local parameters, reside in Property Group subnodes. There are three different types of Property Group nodes:

- The **Basic** group is always present and cannot be removed. To this group you can add properties which are not associated with one particular material model. It is initially empty.
- Material model groups are predefined and contain a fixed set of properties which parameterize a specific material model used by one or more physics interfaces.
- User-defined property groups are in operation identical to the **Basic** group. They can be used for grouping basic properties for example so that different sets of property definitions can be enabled or disabled.

You can add material properties to a material in three different ways. The quickest way is often to enter an expression for a missing but required property in the **Value** column in the **Material Contents** section. This will automatically add the property, and if necessary the property group to which it belongs.

To explicitly add a property, expand the **Material Properties** section and browse the tree of available material property categories (**Basic Properties**, **Acoustics**, and so on). Select a material property leaf or a property group ( ). 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.

	Adding a property belonging to a property group will add the entire group. If you do not need some of the properties in the group, you can leave them undefined.

	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 the **Material** and directly in the **Property Group**.

- When material properties are added from the **Basic** node's or a user-defined group node's **Settings** window, 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** contains all material properties whether they were added from the material node or from a subnode.

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 can exhibit anisotropy induced by external fields. In practice, this means that any anisotropic tensor properties in a nonsolid material must be functions of some external vector field.

MATERIAL CONTENTS

This section lists all output properties declared by the material, or required from the material by the physics in the model. The table lists the **Property**, **Variable**, **Value**, and **Unit** for each property as well as the **Property group** subnode containing the definition. You can edit property values directly in the **Value** column. This will update the definition in the corresponding subnode.

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 either to the Basic property group or to a corresponding material model group, which is automatically created if missing.
- A warning sign () indicates that the material property has been added so that it is declared by the material but is still undefined. An entry is only required if the material property is to be used in the model.
- A check mark () indicates that the property has a defining **Value** and is currently being used in the physics of the model.
- A synchronize symbol () indicates that the property is computed and synchronized using the given values for other material properties from which it can be computed.
- 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 that is not synchronized by editing its value directly in the **Value** column, or, for a selected property, click the **Edit** button () or right-click and choose **Edit** 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 **Full**. The **Variable** column lists the variable names corresponding to the degree of anisotropy. For example, for a symmetric electric conductivity, it contains **{sigma11, sigma12, sigma22, sigma13, sigma23, sigma 33}; sigmaij = sigmaji**. For an isotropic electric conductivity, it contains **sigma_iso; sigmaii = sigma_iso, sigmaij = 0**, where `sigma_iso` is the name of the variable for the isotropic electric conductivity (available as, for example, `mat1.def.sigma_iso`).

If a material has properties that are defined using synchronization rules, the row for that property in the table will be disabled. To break the synchronization and add a value for this property, select the disabled row in table and click the **Break Synchronization** button ( ) in the toolbar underneath the table. The material model will then be added automatically, and the values of the parameters in the material model will be set to the expression used by the synchronization. It is then also possible to edit the expression for the property.

You can right-click any defined material property in the table and choose **Go to Requesting Node**; the focus in the model tree then moves to the node that requested that material property. If there is more than one node that requests that material property, a **Go to Requesting Node** dialog appears, where you can choose which of those nodes to go to. Also, for nodes such as **Material Link** and **Layered Material Link**, you can right-click any defined material property and choose **Go to Source** to move to the material node that is the source of that material property.

MATERIAL LIBRARY SETTINGS

Select the **Material Library Settings** checkbox in the **Show More Options** dialog to display this section for **Material** nodes under **Global Definitions**.

In this section, you can define properties for materials in a user-defined material library to create a set of materials with varying material phase and orientation or variation.

Click the **Update the Label** button ( ) to update the label, which becomes the name of the material in the user-defined material library.

From the **Phase** list, choose **Custom**, **Solid**, **Liquid**, or **Gas** to define the material's phase. This setting is not available if the parent **Group** node has its **Group type** set to **Phase and orientation**, in which case the phase is defined in that **Group** node's settings.

In the **Orientation/variation** field, enter some orientation or variation that represent this material.

Under **Material information**, it is possible to add material information such as a DIN or SAE number or using a custom title with some information. The type or title appears in the **Title** column. All entries appear with their information in the **Text** column. Use the **Move Up** ( ), **Move Down** ( ), and **Delete** ( ) buttons to rearrange the material information in the table, and click the **Add** ( ) button to add more material information.

Click the **Edit Material Information** ( ) button to edit the content of the selected row. You can only add one table row for each type except **Custom**. The following material identification types are available from the **Type** list:

- **DIN**: The Deutsche Industrie Norm (DIN), a German standard.
- **UNS**: The unified numbering system (UNS) is an alloy designation system widely accepted in North America.
- **WNr**: The Werkstoffnummer (WNr) is a German material numbering system.
- **ISO/EN/DIN**: Numbering common to the ISO (International Organization for Standardization), EN (European Standards or Euronorm), and DIN.
- **JIS**: The Japanese Industrial Standards (JIS), a Japanese standard.
- **ASTM**: ASTM International, an international standards organization.
- **AISI**: Designations for materials from the American Iron and Steel Institute (AISI).
- **SAE**: Designations for materials from the Society of Automotive Engineers (SAE).
- **AMS**: SAE aerospace material specifications (AMS).
- **AFNOR**: Association Française de Normalisation (AFNOR) is a Paris-based standards organization and a member body for France at the International Organization for Standardization.
- **Custom** (the default): Add any custom material information type. For **Custom**, also add a **Title** for the information type.

For all entries, add the desired text in the **Information** field.

Click **OK** to save the edits. This material information then appears above the **Properties** table when you select the material from a library in the **Material Browser** window, and you can search using, for example, a material identification number.

APPEARANCE

The settings in this section, available for materials in a component, 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 **Material type** list provides quick settings approximating the appearance of a number of materials — **Air**, **Aluminum**, **Brick**, **Concrete**, **Copper**, **Glass**, **Gold**, **Iron**, **Lead**, **Magnesium**, **Oil**, **Plastic**, **Rock**, **Soil**, **Steel**, **Titanium**, **Water**, **Wood**, and many more. You can also choose **Default geometry material** and **Default plot material**, which can be useful to, for example, use in surface plots in apps to show something that looks like a geometry.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 **Material type** list. If you have chosen a family other than **Custom**, click the **Customize** button to define a custom material. For **Plastic**, **Default plot material**, and some other materials, a separate **Color** list is available for adding a color (default: white). For **Default geometry material** the color is taken from the color theme.

Specular Color, Diffuse Color, and Ambient Color

For each of these properties, select a standard color from the list: **Black**, **Blue**, **Cyan**, **Gray**, **Green**, **Magenta**, **Red**, **White**, or **Yellow**, or choose **Custom** button to define a custom color from the color palette that becomes available underneath the list of colors.

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*.

Custom Basis for Brush Lines

Select the **Custom basis for brush lines** checkbox to define a custom coordinate basis for the brush lines. The default values for the brush line axes represent the global Cartesian coordinate system. Specify other x, y, and z coordinates for the **Origin** and the **xm-axis** if desired. To specify other coordinates for the **ym-axis**, first select the **Specify ym-axis** checkbox.

Normal Mapping

Select the **Normal mapping** checkbox to add normal vector noise and brush lines. Normal mapping is a texture that disturbs the normals when calculating lighting on the surface (also called bump mapping). This causes the surface to look rough and textured. The default **Normal vector noise scale** and **Normal vector noise frequency** are taken from the material. You can choose the type from the **Noise type** list: **White noise** (a uniform distribution) or **Simplex noise**.

Enter other values as needed, or click to clear the **Normal mapping** checkbox.

- The **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.
- The **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.
- Add brush lines if desired by choosing an option from the **Brush lines** list: **No brush lines** (the default), **Brush lines orthogonal to xm-axis**, **Brush lines orthogonal to ym-axis**, **Brush lines orthogonal to zm-axis**, **Brush lines along xm-axis**, **Brush lines along ym-axis**, **Brush lines along zm-axis**, **Brush lines around xm-axis**, **Brush lines around ym-axis**, **Brush lines around zm-axis**, or **Brush lines around origin**.

The "m" in **xm**, **ym**, and **zm** above represents that the coordinates are with respect to the material basis (custom basis).

Additional Color

Select the **Additional color** checkbox if you want to blend the appearance with an additional color that is added on the surface using noise. The settings are similar to those for the normal mapping above, but you can also choose the color to add from the **Color** list and choose a **Color blend** between 0 and 1. The **Noise color** is a parameter that affects the appearance of the added color.

Opacity

The default **Opacity** is 1.

Lighting Model

The default **Lighting model** — **Blinn–Phong** or **Cook–Torrance** — is based on the material. Select **Simple** instead as needed; it has no additional settings.

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 settings 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, and it includes the following settings.

- The **Reflectance at normal incidence** value is the amount of incoming light from the normal direction (of the surface) that is reflected.
- The **Surface roughness** is a value that describes microreflectance on the surface. Higher values create a rougher look of the surface with fewer highlights.
- The **Anisotropy** is a value that defines the anisotropic specular highlighting. You can use it to give an effect that could be the result of machining or polishing in a certain direction. Select the **Flip anisotropy** checkbox to flip that apparent direction. Also, enter the direction of the anisotropy axis as a vector with three components in the **Anisotropy axis** field.
- The **Metallic** parameter is a value that affects how metallic the material appears to be. A less metallic material reflects less of the environment. A nonmetallic material cannot reflect the environment
- The **Pearl** parameter is an artificial effect that mimics the colors of a pearl.
- The **Diffuse wrap** value gives an artificial effect that you can use to emulate subsurface scattering.

- The **Clear coat** parameter is an artificial effect that makes surfaces that are parallel to the view direction more white. It also adds specular highlighting and reduces the effect on the environment reflections from the normal mapping.
- The **Reflectance** value is an additional setting that affects how much of the environment that is reflected.

You can enter these values in the respective text fields; alternatively, use the sliders underneath each text field to adjust those values between 0 and 1. If you use the sliders, the material appearance is updated directly.

Material Library Link

You can add **Material Library Link** nodes () from the **Add Materials** and **Material Browser** windows or by choosing **Add Material from Library** in a **Material Link** node. A **Material Library Link** provides a link to a material from a material library, which can easily be updated when material properties change in the material library file. It works just like any other global or local material when added to the model. For most settings, see The Settings Window for Material. When the link is active, you cannot change any of the values in the **Material Contents** section. You can also not add any user-defined property groups.

MATERIAL LIBRARY

At the top of this section, the material that is linked and the material library from which it has been added appear.

Click the **Reload** button to reload the material properties for the linked material from its material library, if the material properties have changed in the library material. If you have broken the link, you will get a question about if you want to override all changes.

Click the **Replace Material** button to open a **Replace Material in Link** window, where you can choose another material from the available material libraries. Click the **Replace Material in Link** button to replace the material in the **Material Library Link** node and close the **Replace Material in Link** window.

Click the **Break Link** button to break the link to the material library. It can be necessary if the library material does not include all material properties needed for a simulation. The **Material Library Link** node then behaves like a **Material** node, and you change the values of material properties in the **Material Contents** section and add user-defined property groups.

Property Groups

The **Property Group** nodes ( ) define output properties and declare model inputs on which the property values can depend. There is a slight difference in the **Settings** window between, on one hand, the **Basic** and user-defined property groups, and on the other hand the built-in material model groups. The latter do not allow adding additional output properties, but all property groups allow declaring model inputs and also add local properties which can be used in output property expressions.

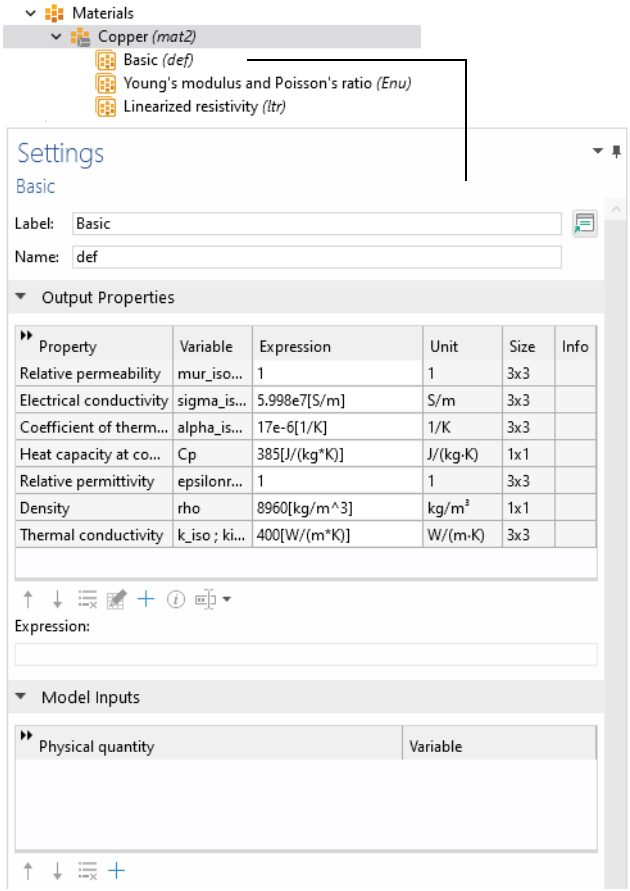


Figure 2-5: An example of the default Basic property group's Settings window.

In the **Label** field, enter a suitable label for the property group if needed. Changing the **Name** field changes the namespace identifier used when defining output property variables and functions. You can, for example, make the identifier shorter.Changing the name of a user-defined property group will also update its tag. Other property group types have fixed tags.

OUTPUT PROPERTIES

The currently declared 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 **Select Quantity** button (+) to add another output property. Select the desired quantity from the tree in the **Physical Quantity** dialog that opens.

If required, edit the expressions in the **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 for easier specification of orthotropic and anisotropic material properties (tensors), when applicable. Select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** when entering the data in the material property's dialog. 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 output 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 electric 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 ( ), enter the property information in the dialog that opens, and then click **OK**. When information is available for a property, an information symbol ( ) appears in the **Info** column.

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

Output Property Variables

The **Basic** property group under a material, as well as any user-defined group, creates the following variables:

TABLE 2-2: VARIABLES GENERATED FROM THE BASIC PROPERTY GROUP AND USER-DEFINED GROUPS.

VARIABLE NAME	SCOPE	SELECTION	EXAMPLE
Variable name of physical quantity	`root.material`	Material selection	`root.material.rho`
	`root.<comp>.<mat>.<group>`	Global selection	`root.comp1.mat1.def.rho`
Physical quantity name	`root.<comp>.<mat>.<group>`	Global selection	`root.comp1.mat1.def.density`

Predefined material model property groups define the following variables:

TABLE 2-3: VARIABLES GENERATED FROM THE BASIC PROPERTY GROUP AND USER-DEFINED GROUPS.

VARIABLE NAME	SCOPE	SELECTION	EXAMPLE
Property name	`root.material.group`	Material selection	`root.material.linzRes.alpha`
	`root.<comp>.<mat>.<group>`	Global selection	`root.comp1.mat1.linzRes.alpha`

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 makes the variable name `T` available for use in 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 in the example above only works if there is a variable called `T` available on geometric entities where the material is used.

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

Use the **Move Up** (↑), **Move Down** (↓), and **Delete** ( ) buttons to organize the table as needed.

LOCAL PROPERTIES

Here you can enter user-defined properties which are local to the property group. This means that they are accessed from physics interfaces and do not generate variables in the `material.` scope. These local properties become available for use in the property group's output property expressions, and can also be accessed more generally using the full property group scope.

Enter a name in the **Name** column and its definition in the **Expression** column. You can also enter a **Description**. Information about all local properties from all property groups is also displayed in the **Material Contents** section of the parent **Material** node. In that node, only the **Expression** column is editable.

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 (in 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 in 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. To add it, right-click the **Materials** node and choose **Material Link** from the **More Materials** submenu. 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 or material library link 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 **Material Switch** node () to switch between materials during a solver sweep. You add the materials as subnodes under the **Material 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. You can also add a **Layered Material** or a **Single Layer Material**.

The switch for materials essentially acts 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 parameter name is constructed based on the tag of the Material Switch node, using the special namespace `matsw`. For example, the parameter controlling a Switch node on the global level will typically be `matsw.sw1`, while for a component-level Switch it will be `matsw.comp1.sw1`.

During a material sweep, the sweep parameter takes consecutive integer values, starting from one, indicating which material under the switch that should currently provide material properties. You can use the parameter name in conditional expressions to control also other aspects of the model. Conversely, it is possible to control a material Switch also by manually defining the full switch parameter name in a **Parameters** node. You can then choose the parameter to sweep over in a standard **Parametric Sweep** node or assign it different (integer) values in different parameter **Case** nodes and sweep using a *Parameter switch* sweep.

The **Material 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 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).
- **Linear Elastic Material, Layered** in the Shell or Membrane interface (requires the Composite Materials Module).
- **Hyperelastic Material, Layered** in the Shell interface (requires the Composite Materials Module).
- **Piezoelectric Material, Layered** 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). The thickness can be numeric value or a scalar parameter.

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 that are explicitly referenced by physics features. The default is to take the material **From layer**. The interface material properties are then computed from the adjacent layers’ material properties.

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.

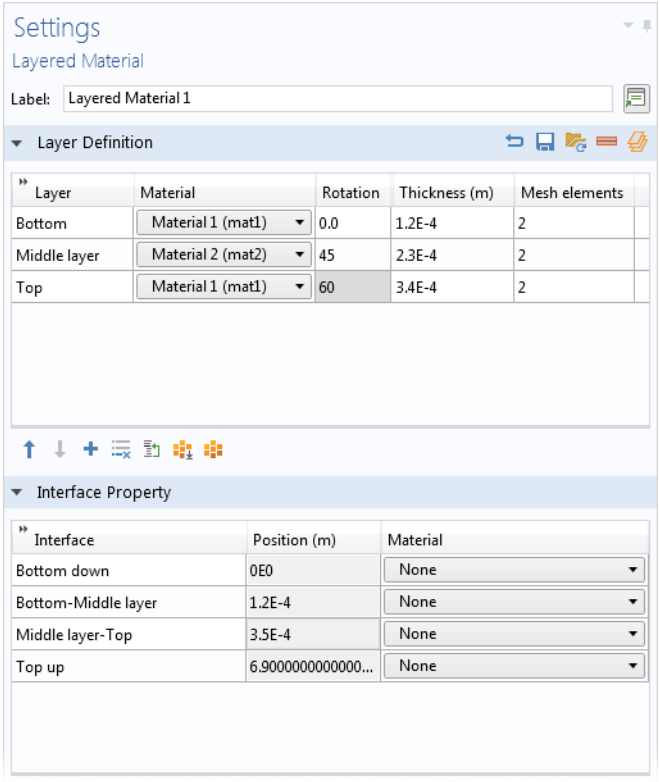


Figure 2-6: Settings for a material with three layers.

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.

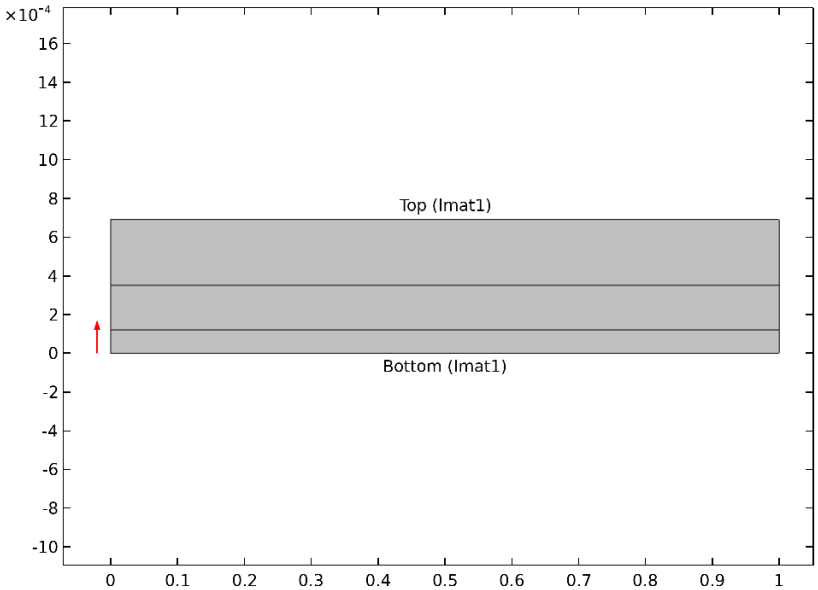


Figure 2-7: The layer cross section plot for a material with three layers.

You can also click the downward pointing arrow to choose **Layer Cross Section Preview** ( ) or **Create Layer Cross Section Preview** ( ), which adds the preview plot as a new plot group under **Results**. 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 2-10, but there is only a single layer.

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. You can also click the downward pointing arrow to choose **Layer Stack Preview** ( ) or **Create Layer Stack Preview** ( ), which adds the preview plot as a new plot group under **Results**. Click the **Create Layer Stack Plot** button ( ) to add the preview of the layer stack as a new plot group under **Results**.

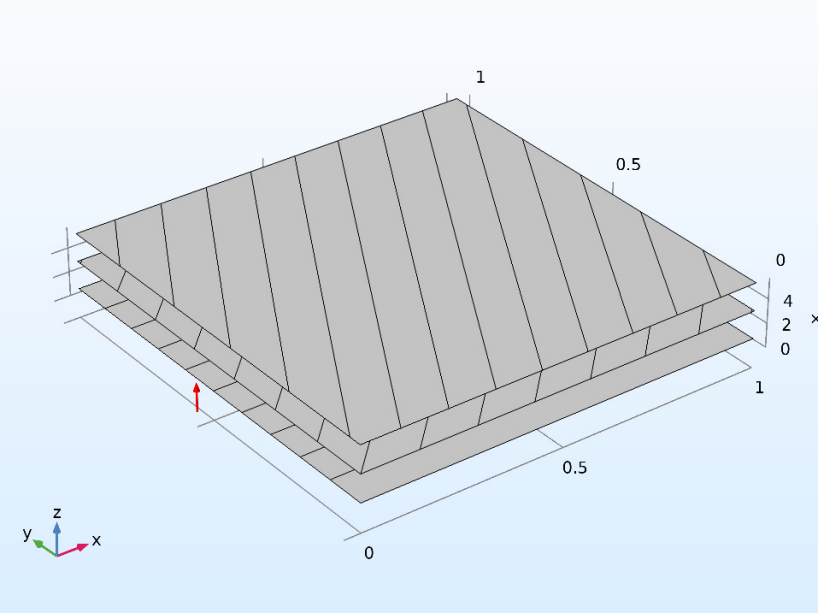


Figure 2-8: The layer stack preview plot for a material with three layers.

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 clear the corresponding checkbox, 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** checkbox to remove the text labels showing layer names and materials.

Layered Material Link

The **Layered Material Link** node ( ) 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.

LAYERED MATERIAL SETTINGS

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.

From the **Transform** list, choose one of the following options:

- **None** (the default), for no transformation.
- **Symmetric** or **Antisymmetric**, to create a symmetric or antisymmetric layered material when the information of layers of one side of the midplane is supplied. Choose which side to mirror in from the **Mirror in** list: **Upside** (the default) or **Downside**. Upside means that the symmetry layers are on the top of the original layers. The symmetry line is the top-side boundary. Downside means that the symmetry layers are on the bottom of the original layers. The symmetry line is the bottom-side boundary. Select the **Merge middle layers** checkbox to merge the two middle layers into one to create an odd symmetric layer.
- **Repeated**, to create a number of repeating stacks, which you enter in the **Number of repeats** field (default: 1).

Select the **Scale** checkbox to scale the layered material's thickness with a factor (default: 1).The scale can be a numerical value, a parameter, or an expression. Such an expression can, for example, be a function of the coordinates so that a surface with variable thickness can be described.

If a single layer in a laminate has a variable thickness, you can define that layer in either a separate **Layered Material** or in a **Single Layer Material**.

- When using a **Layered Material**, apply the scaling expression in a **Layered Material Link**, and then use a **Layered Material Stack** to build the complete laminate.
- When using a **Single Layer Material** with an expression for the thickness, use a **Layered Material Stack** to build the complete laminate.

If you have defined a layer with a scaling factor, it appears in the preview window with a darker color than a nonscaled layer.

The preview is not shown in the base geometry space, so it will not show any geometrical dependency.

The labels of the newly created layers include a suffix to distinguish them from the original layers:

- **(sym)** for the symmetric layers.
- **(asym)** for the antisymmetric layers.
- **(repX)** for the repeated layers (number *X*).

Click the **Layer Cross Section Preview** button (to plot a preview of the layer cross section including the transformation (see the following plot for an example).

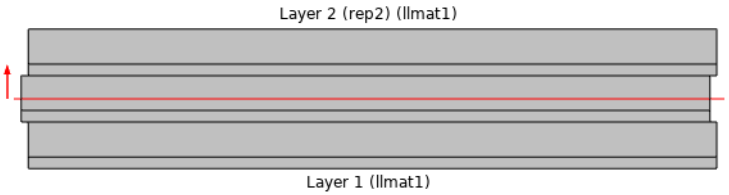


Figure 2-9: A repeat laminated stacks with 2 times repeated layers.

You can also click the downward pointing arrow to choose **Layer Cross Section Preview** ( ) or **Create Layer Cross Section Preview** ( ), which adds the preview plot as a new plot group under **Results**.

Click the **Layer Stack Preview** button ( ) to get a preview of the stack with the transformation.

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 can 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 layered material, including the location of the reference surface (Figure 2-10). 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. You can also click the downward pointing arrow to choose **Layer Cross Section Preview** ( ) or **Create Layer Cross Section Preview** ( ), which adds the preview plot as a new plot group under **Results**.

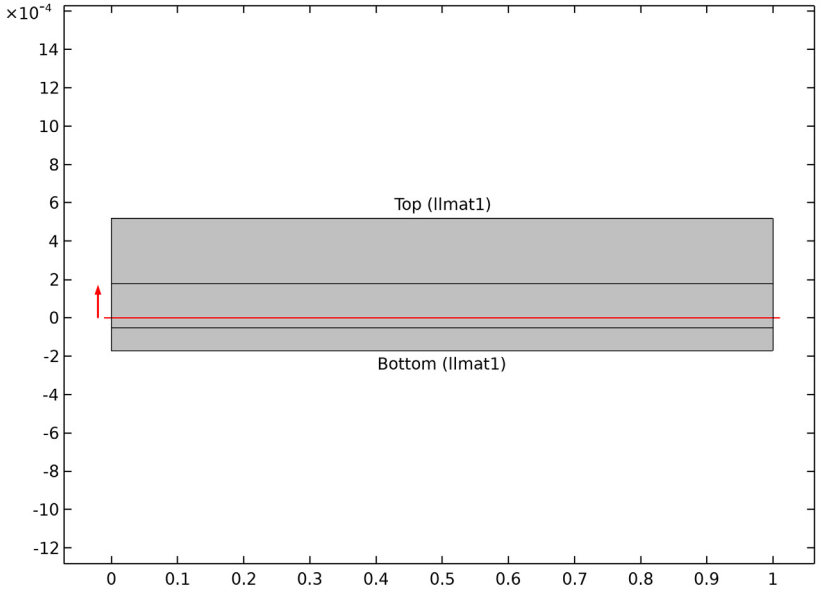


Figure 2-10: Layer cross section preview plot with relative offset set to 0.5.

NONLAYERED MATERIAL SETTINGS

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. The default settings is **Same as layered material**, which means that the nonlayered material properties are computed as an average value of the layer's material properties. 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.

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** checkbox 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 Stack

In the **Layered Material Stack** node ( ), you can compose a new layered material by stacking other layered materials (including Material nodes that define single-layer 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.

Ply modeling can be achieved from the selection of substack materials. By using different selections on each stack member, you can create ply-based selections. From the ply-based selections, a set of the product selections, called zone-based selections, can be deduced and displayed in a **Stack Zone Definition** section (see below).

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.

The interface between the two **Layered Material Stack** nodes takes the interface material from the first **Layered Material Stack** node and ignores the interface material of the second **Layered Material Stack** node.

LAYERED MATERIAL SETTINGS

From the **Transform** list, choose one of the following options:

- **None** (the default), for no transformation.
- **Symmetric** or **Antisymmetric**, to create a symmetric or antisymmetric layered material when the information of layers of one side of the midplane is supplied. Choose which side to mirror in from the **Mirror in** list: **Upside** (the default) or **Downside**. Upside means that the symmetry layers are on the top of the original layers. The symmetry line is the top-side boundary. Downside means that the symmetry layers are on the bottom of the original layers. The symmetry line is the bottom-side boundary. Select the **Merge middle layers** checkbox to merge the two middle layers into one to create an odd symmetric layer.
- **Repeated**, to create a number of repeating stacks, which you enter in the **Number of repeats** field (default: 1).

Select the **Scale** checkbox to scale the layered material's thickness with a factor (default: 1). If you have defined a layer with a scaling factor, it appears in the preview window with a darker color than a nonscaled layer.

The preview is not shown in the base geometry space, so it will not show any geometrical dependency.

The labels of the newly created layers include a suffix to distinguish them from the original layers:

- **(sym)** for the symmetric layers.
- **(asym)** for the antisymmetric layers.
- **(repX)** for the repeated layers (number *X*).

Click the **Layer Cross Section Preview** button ( ) to plot a preview of the layer cross section including the transformation. You can also click the downward pointing arrow to choose **Layer Cross Section Preview** ( ) or **Create Layer Cross Section Preview** ( ), which adds the preview plot as a new plot group under **Results**. Click the **Layer Stack Preview** button ( ) to get a preview of the stack with the transformation. Click the **Create Layer Stack Plot** button ( ) to add the preview of the layer stack as a new plot group under **Results**.

A combination of transformations can be made by defining the transformations for both the **Layered Material Stack** node and a **Layered Material Link** subnode.

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. If you use ply modeling, these additional positions are available: **Midplane of stack member on boundary**, **Downside of stack member on boundary**, **Upside of stack member on boundary**. If you choose one of those positions, also choose a member of the ply stack from the **Stack member** list. Also, with one of those positions, the **Automatic alignment when the selected stack member is not available** checkbox is selected by default. The stack members (in zones that do not have the selected stack member) will then be aligned with the stack members in the zone that have the selected stack member. Clear it if you do not want that automatic alignment.

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 2-11, 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.

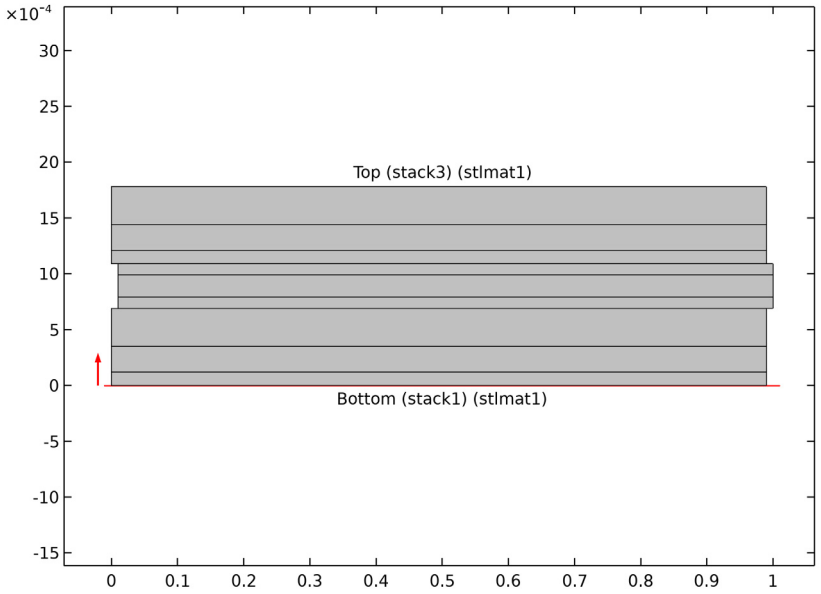


Figure 2-11: Layer cross section preview plot with relative offset set to Down side on boundary.

You can also click the downward pointing arrow to choose **Layer Cross Section Preview** ( ) or **Create Layer Cross Section Preview** ( ), which adds the preview plot as a new plot group under **Results**.

STACK ZONE DEFINITION

This section is available if there are multiple zones. It then lists the names of the zones, the stack members for each zone, and the boundary selections for each zone.

NONLAYERED MATERIAL SETTINGS

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. The default settings is **Same as layered material**, which means that the nonlayered material properties are computed as an average value of the layer's material properties. 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** checkbox 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.

From the **Transform** list, choose one of the following options:

- **None** (the default), for no transform.
- **Symmetric** or **Antisymmetric**, to create a symmetric or antisymmetric layered material when the information of layers of one side of the midplane is supplied. Choose which side to mirror in from the **Mirror in** list: **Upside** (the default) or **Downside**. Upside means that the symmetry layers are on the top of the original layers. The symmetry line is the top-side boundary. Downside means that the symmetry layers are on the bottom of the original layers. The symmetry line is the bottom-side boundary. Select the **Merge middle layers** checkbox to merge the two middle layers into one to create an odd symmetric layer.
- **Repeated**, to create a number of repeating stacks, which you enter in the **Number of repeats** field (default: 1).

Select the **Scale** checkbox to scale the layered material's thickness with a factor (default: 1). The scale can be a numerical value, a parameter, or an expression. Such an

expression can, for example, be a function of the coordinates so that a surface with variable thickness can be described.

If a single layer in a laminate has a variable thickness, you can define that layer in either a separate **Layered Material** or in a **Single Layer Material**.

- When using a **Layered Material**, apply the scaling expression in a **Layered Material Link**, and then use a **Layered Material Stack** to build the complete laminate.
- When using a **Single Layer Material** with an expression for the thickness, use a **Layered Material Stack** to build the complete laminate.

If you have defined a layer with a scaling factor, it appears in the preview window with a darker color than a nonscaled layer.

The preview is not shown in the base geometry space, so it will not show any geometrical dependency.

The labels of the newly created layers include a suffix to distinguish them from the original layers:

- **(sym)** for the symmetric layers.
- **(asym)** for the antisymmetric layers.
- **(repX)** for the repeated layers (number *X*).

Click the **Layer Cross Section Preview** button ( ) to plot a preview of the layer cross section including the transform. You can also click the downward pointing arrow to choose **Layer Cross Section Preview** ( ) or **Create Layer Cross Section Preview** ( ), which adds the preview plot as a new plot group under **Results**. Click the **Layer Stack Preview** button ( ) to get a preview of the stack with the transform. Click the **Create Layer Stack Plot** button ( ) to add the preview of the stack with the transform as a new plot group under **Results**.

Single Layer Materials

To add a single-layer material, choose **Single Layer Material** ( ) from the global **Materials** node's context menu or the **Layers** submenu on the context menu of a **Materials** node in a component. Then, a **Material** node is created with some additional settings (see The Settings Window for Material) and a **Shell** property group (see Shell

Geometric Properties) with a default thickness of 10^{-4} m. You can also switch an **Material** node into a single-layer material by adding a **Shell** property group and define a thickness, and it can also turn into a single-layer material when you specify a value for the requested thickness in the **Material Contents** table, which appears when a layered shell feature requests the material properties from a standard material. The thickness for a single-layer material can be defined as a numerical value, a parameter, or an expression. Such an expression can, for example, be a function of the coordinates so that a surface with variable thickness can be described.

If a single layer in a laminate has a variable thickness, you can define that layer in either a separate **Layered Material** or in a **Single Layer Material**.

- When using a **Layered Material**, apply the scaling expression in a **Layered Material Link**, and then use a **Layered Material Stack** to build the complete laminate.
- When using a **Single Layer Material** with an expression for the thickness, use a **Layered Material Stack** to build the complete laminate.

Single-layer materials provide a quick way to define data for a nonlayered 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**. A single-layer material can be linked by a Layered Material Link, and it can also be a stack member of a Layered Material Stack or a switch member of a Switch for Materials.

Porous Material

Add a **Porous Material** node ( ) under a **Materials** node in a model component to add a definition of a porous material. In the settings for this node you can define the porosity of the porous material, or you can right-click the **Porous Material** node to add Fluid, Solid, Pellet, and Immobile Fluid subnodes as required and define the porous material properties in those subnodes.

POROSITY

If you have not added any subnode for defining the porous material, enter a value or expression between 0 and 1 in the ε_p field. Otherwise, depending on the added subnodes, the corresponding expression for the porosity of the porous material appears here under **Volume fractions of immobile phases**.

PHASE-SPECIFIC PROPERTIES

Physics features that support porous media request properties per phase present. These properties are listed in the **Phase-Specific Properties** table. All phase-specific properties need to be specified in a corresponding subnode to the **Porous Material** node (as a **Solid**, **Liquid**, or **Pellet** subnode). The list can contain multiple entries of the same kind, such as the density of the fluid and the density of the solid. When a subnode is missing, a red stop symbol appears in the table, and the **Add Required Phase Nodes** button () is active. Click that button to create the missing subnodes. You then need to go to the created subnodes and define all requested properties there.

Right-click a row in the Property and Phase columns to choose **Go to Requesting Node** to move to the physics node that requested the property, or choose **Go to Source** to move to the subnode that is the source for the property, when applicable.

Click any column header to sort the list alphabetically according to the content in that column. You can use this to, for example, list the properties according to their phase.

HOMOGENIZED MATERIAL

From the **Material** list, choose the material to represent the homogenized material: **Locally defined** (the default), or select any other material that you have added. Click the **Go to Material** button () to move to the selected material's node. Click the **Add Material from Library** button ( to add a material from the **Add Material to Pellet** window. You can also click the downward arrow next to the button and choose **Blank Material** to add a new blank material. In both cases, the added material becomes selected in the **Material** list.

HOMOGENIZED PROPERTIES

When physics features that do not support porous media take properties **From material**, and the domain material is a porous material, then properties are shown in the **Homogenized Properties** table. Here, there is only one set of properties and not one per phase. The properties are to be thought of as homogenized in some manner. The material is porous, but for the one-phase interface, homogenized properties are used. In this manner you can, for example, add a Solid Mechanics interface to a model of porous media and model the structural mechanics part as usual. When the **Material** in the **Homogenized Material** section is **Locally defined**, enter values or expressions for the properties in the **Value** column.

APPEARANCE

For the **Appearance** settings, see Appearance.

Fluid

Right-click a **Porous Material** node to add a **Fluid** subnode ( ) to define properties of a fluid phase in the porous medium.

FLUID PROPERTIES

From the **Material** list, choose the material to represent the fluid: **Locally defined** (the default), or select any other material that you have added. Click the **Go to Material** button ( ) to move to the selected material's node. Click the **Add Material from Library** button (+ to add a material from the **Add Material to Pellet** window. You can also click the downward arrow next to the button and choose **Blank Material** to add a new blank material. In both cases, the added material becomes selected in the **Material** list.

MATERIAL PROPERTIES

This section is only available if you have selected **Locally defined** from the **Material** list in the **Fluid Properties** section. You can then choose material properties that you want to add for the fluid from the list and then click the **Add to Material** button (+) to add those properties to the table in the **Material Contents** section below.

MATERIAL CONTENTS

This section contains a table with the material properties for the fluid, which typically consist of the porosity and the material properties from the chosen material for the fluid. When the **Material** in the **Fluid Properties** section is **Locally defined**, enter values or expressions for the properties in the **Value** column. You can also click the **Edit** button ( ) below the table to enter the property values.

Solid

Right-click a **Porous Material** node to add a **Solid** subnode ( ) to define properties of a solid phase in the porous medium.

SOLID PROPERTIES

From the **Material** list, choose the material to represent the solid: **Locally defined** (the default), or select any other material that you have added. Click the **Go to Material** button ( ) to move to the selected material's node. Click the **Add Material from Library** button (+ to add a material from the **Add Material to Pellet** window. You can also click the downward arrow next to the button and choose **Blank Material** to add a

new blank material. In both cases, the added material becomes selected in the **Material** list.

For the **Volume fraction** of the solid, enter a value between 0 and 1 in the θ_s field.

MATERIAL PROPERTIES

This section is only available if you have selected **Locally defined** from the **Material** list in the **Solid Properties** section. You can then choose material properties that you want to add for the solid from the list and then click the **Add to Material** button (+) to add those properties to the table in the **Material Contents** section below.

MATERIAL CONTENTS

This section contains a table with the material properties for the solid, which typically consist of the porosity and the material properties from the chosen material for the solid. When the **Material** in the **Solid Properties** section is **Locally defined**, enter values or expressions for the properties in the **Value** column. You can also click the **Edit** button ( ) below the table to enter the property values.

Pellet

Right-click a **Porous Material** node to add a **Pellet** subnode ( ) to define properties of a pellet phase in the porous medium. The **Pellet** subnode provides the properties for a certain type of pellets that the solid part of the porous material consists of. A porous material can include pellets of different types. For example, the pellets might be a mix of 30% spherical pellets with a certain diameter, 20% spherical pellets with another diameter, and 50% of a third type of pellet that is cylindrical instead of spherical. Those 100% together form the solid phase in the "pellet bed".

A **Pellet** node provides the geometrical properties for pellets and material properties for the solid phase in the pellet, such as density and heat capacity. The material properties for the fluid phase should be defined in a Fluid subnode.

When a **Porous Material** node is used in the physics, then all pellet types will exist at each point in the geometry. Therefore, the plural form, **Pellets** as a subnode to the **Packed Bed** node, used in the Chemical Reaction Engineering Module to model packed bed reactors with catalytic pellets. Also, **Pellets** as a subnode to the **Porous Medium** node can be used to model the local temperature field within pellets when using a Heat Transfer in Porous Media interface.

PELLET PROPERTIES

From the **Material** list, choose the material to represent the pellet: **Locally defined** (the default), or select any other material that you have added. Click the **Go to Material** button ( ) to move to the selected material's node. Click the **Add Material from Library** button (+) to add a material from the **Add Material to Pellet** window. You can also click the downward arrow next to the button and choose **Blank Material** to add a new blank material. In both cases, the added material becomes selected in the **Material** list.

From the **Shape** list, define the shape of the pellet: **Sphere** (the default), **Cylinder**, **Flake**, or **User defined**:

- For **Sphere**, further specify a **Diameter** $d_{\text{pe,s}}$ (SI unit: m), where the default value is 1 mm. The effective diameter $d_{\text{pe,eff}}$ is then equal to $d_{\text{pe,s}}$.
- For **Cylinder**, further specify a **Diameter** $d_{\text{pe,c}}$ (SI unit: m), where the default value is 1 mm.The effective diameter $d_{\text{pe,eff}}$ is then equal to $\frac{3d_{\text{pe,c}}}{2}$.
- For **Flake**, define a **Thickness** $w_{\text{pe,f}}$ (SI unit: m), where the default value is 1 mm, in a similar way. The effective diameter $d_{\text{pe,eff}}$ is then equal to $3w_{\text{pe,f}}$.
- For **User defined**, specify the **Pellet surface** A_{pe} (SI unit: m^2) and **Pellet volume** V_{pe} (SI unit: m^3). The default values are 1 m^2 and 1 m^3, respectively. The effective diameter $d_{\text{pe,eff}}$ is then equal to $\frac{6V_{\text{pe}}}{A_{\text{pe}}}$

Also specify a **Porosity** ε_{pe} as a relative number between 0 and 1 (default: 0.1). This is the microscale porosity of the inside of an individual pellet.

If you define a second **Pellet** subnode, the **Pellet Bed Properties** section below is not available. Instead, specify a **Bed volume fraction**, a relative number between 0 and 1. Only the first **Pellet** subnode can define the pellet bed porosity. For additional **Pellet** subnodes, you instead specify the volume fraction relative to the other pellet types.

PELLET BED PROPERTIES

In this section you specify the porosity of the pellet bed. From the **Porosity** list, choose **User defined** (the default) or **From densities**. With **User defined**, add a value for the porosity ε_{p} as a relative number between 0 and 1 (default: 0.26, representing perfectly packed spheres). With **From densities**, the porosity is defined as $1 - \rho_{\text{b}}/\rho_{\text{pe}}$, where you define the density of the packed bed, ρ_{b}, in the **Density** field (default: 800 kg/m^3).

MESH

In this section you specify the properties of the mesh for the extra dimension that each **Pellet** subnode defines so that both the species concentration and the temperature can be computed locally in the pellet.

For the mesh distribution, choose **Linear**, **Cubic root sequence** (the default), or **Square root sequence** from the **Distribution** list. Also, specify the **Number of elements** (default: 6).

Click the **Extra Dimension Mesh Preview** button ( ) in the **Mesh** section toolbar to get a plot of the extra dimension mesh in a separate plot window, showing the mesh distribution and direction.

MATERIAL PROPERTIES

This section is only available if you have selected **Locally defined** from the **Material** list in the **Pellet Properties** section. You can then choose material properties that you want to add for the pellet from the list and then click the **Add to Material** button (+) to add those properties to the table in the **Material Contents** section below.

MATERIAL CONTENTS

The table in this section contains the material properties for the pellet material. When the **Material** in the **Pellet Properties** section is **Locally defined**, enter values or expressions for the properties in the **Value** column. You can also click the **Edit** button ( ) below the table to enter the property values.

Immobile Fluid

Right-click a **Porous Material** node to add a **Immobile Fluid** subnode ( ) to define properties of an immobile fluid phase in the porous medium.

IMMOBILE FLUID PROPERTIES

From the **Material** list, choose the material to represent the immobile fluid: **Locally defined** (the default), or select any other material that you have added. Click the **Go to Material** button ( ) to move to the selected material's node. Click the **Add Material from Library** button (+ to add a material from the **Add Material to Pellet** window. You can also click the downward arrow next to the button and choose **Blank Material** to add a new blank material. In both cases, the added material becomes selected in the **Material** list.

For the immobile fluid's **Volume fraction**, enter a value between 0 and 1 in the θ_{imf} field.

MATERIAL PROPERTIES

This section is only available if you have selected **Locally defined** from the **Material** list in the **Immobile Fluid Properties** section. You can then choose material properties that you want to add for the immobile fluid from the list and then click the **Add to Material** button (+) to add those properties to the table in the **Material Contents** section below.

MATERIAL CONTENTS

This section contains a table with the material properties for the immobile fluid, which typically consist of the porosity and the material properties from the chosen material for the immobile fluid. When the **Material** in the **Immobile Fluid Properties** section is **Locally defined**, enter values or expressions for the properties in the **Value** column. You can also click the **Edit** button ( ) below the table to enter the property values.

Topology Link

Add a **Topology Link** node ( ) under a **Materials** node in a model component to add material based on a material that you have added under the global **Materials** node ( ) and **Density Model** use( ) that you have added under the component **Definitions** node ( ). To add it, right-click the **Materials** node and choose **Topology Link** from the **More Materials** submenu.

The **Topology Link** will be added automatically, if one of the topology optimization study steps is chosen in the model wizard.

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.

From the **Topology source** list, select the **Density Model**, which should modify the material. The Density model will scale the Young's modulus with the *penalized material volume factor* (`dtopo#.theta_p`), and the density with the *output material volume factor* (`dtopo#.theta`).

Multiphase Material

Add a **Multiphase Material** node ( ) under a **Materials** node in a model component to define a material composed of two or more immiscible phases. Instead of resolving the exact position in space of each phase, they are approximated by their volume fraction, and the effective properties of the material are obtained from a set of mixing rules.

The material properties of each phase are defined on their respective Phase (Subnode) subnode.

FRACTIONS

The **Volume fraction definition** can either be **Locally defined** or controlled from a physics interface providing volume fractions, such as the **Level Set**, **Phase Field**, **Ternary Phase Field**, and **Phase Transport** interfaces. The default is **Locally defined**. When the volume fractions are defined locally, they can be specified as user inputs in each Phase (Subnode) subnode or in the phases table described below. The Multiphase Material feature will automatically constrain each volume fraction to be between 0 and 1.

Use the **Constrained phase** setting to control which phase that does not have an explicit definition of its volume fraction but instead gets the remaining volume fraction as 1 minus the sum of fractions from the other phases. The default is **First phase**.

PHASES

The **Phases** table displays a summary of all of the phases in the material, the source of their material properties, and their volume fraction expression. See Phase (Subnode) for more information.

MATERIAL CONTENTS

The material properties in the multiphase material are defined by a mixing rule that depends on the volume fractions and one or more material properties of each phase.

The material content table contains the mixing rules for the material properties for the multiphase material. The mixing rule for a specific material property can be edited in a separate window. To open this window, right-click the row containing the material property and select **Edit Mixing Rule** ( ). You can also select a row and click the **Edit Mixing Rule** button ( ) under the table. The **Edit Mixing Rule** button ( ) cannot be used if more than one row are selected or if no row is selected. See The Edit Mixing Rule Dialog for more information on selecting mixing rules.

When the **Mixing rule** for a material property is set to **From homogenized properties**, enter values or expressions for the properties in the **Homogenized Properties** section.

HOMOGENIZED MATERIAL

From the **Material** list, choose the material to represent the homogenized material properties: **Locally defined** (the default), or select any other material that you have added. Click the **Go to Material** button ( ) to move to the selected material's node. Click the **Add Material from Library** button (+) to add a material from the **Add Material from Library** window. You can also click the downward arrow next to the button and choose **Blank Material** to add a new blank material. In both cases, the added material becomes selected in the **Material** list.

HOMOGENIZED PROPERTIES

Material properties that use **From homogenized properties** instead of a specific mixing rule are shown in the **Homogenized Properties** table. The properties are to be thought of as homogenized in some matter other than through the available mixing rules. When the **Material** in the **Homogenized Material** section is **Locally defined**, enter values or expressions for the properties in the **Value** column. You can also click the **Edit** button ( ) below the table to enter the property values.

APPEARANCE

For the **Appearance** settings, see Appearance.

Effective Material

Add an **Effective Material** node ( ) under a **Materials** node in a model component to define a material composed of two or more immiscible constituents. Instead of resolving the exact position in space of each constituent, they are approximated by their volume fraction, and the effective properties of the material are obtained from a set of mixing rules.

The **Effective Material** node is similar to the Multiphase Material node, with the exception that it contains Constituent (Subnode) instead of **Phases** and that it includes special mixing rules for composite materials.

Phase (Subnode)

Right-click a **Multiphase Material** node to add a **Phase** subnode ( ) to define properties of a phase in the material.

The **Phase** node's **Settings** window is similar to the **Settings** window for a material node (see The Settings Window for Material), with the exception that it includes **Volume Fraction** and **Link Settings** sections:

VOLUME FRACTIONS

When the phase material is selected as the constrained phase in the parent multiphase material, its volume fraction is defined as one minus the sum of volume fractions of the other phases. If the current phase is not the constrained phase, enter a value or expression between 0 and 1 in the **Volume fraction** V_f field.

LINK SETTINGS

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

- **Locally defined** (the default). In this case, the **Material Properties** section will be available (see The Settings Window for Material).
- Any global material node.
- Any material in the component, except the parent multiphase 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.

Constituent (Subnode)

Right-click an **Effective Material** node to add a **Constituent** subnode ( ) to define properties of a constituent in the material.

The **Constituent** subnode's **Settings** window is identical to the **Settings** window for a **Phase** subnode (see Phase (Subnode)).

The Edit Mixing Rule Dialog

The **Edit Mixing Rule** dialog is used to control the mixing rules in the **Material Contents** table of a Multiphase Material or Effective Material.

To open the **Edit Mixing Rule** dialog:

- Right-click the desired row in the **Materials Contents** table and select **Edit Mixing Rule** ( ).
- Select a row and click the **Edit Mixing Rule** button ( ) under the table. The button is not active if more than one row are selected or if no row is selected.

If there are several material properties in the table, you can step through the table without closing the dialog using the **Previous Row (Discard Changes)** ( ), **Previous Row**

(Store Changes) ( ), **Next Row (Discard Changes)** ( ), and **Next Row (Store Changes)** ( ) toolbar buttons. The current row of the is identified by the **Property Group** and **Parameter** fields.

The Multiphase Material and Effective Material features support several different mixing rules, which you choose from the **Mixing rule** list. Some of the rules are only available for specific material parameters. The mixing rules for composites are exclusive of the Effective Material, and the Krieger-type viscosity is only available for the Multiphase Material. See the sections below for more information about the mixing rules.

BASIC MIXING RULES

The basic mixing rules are available to all material properties.

Volume Average

The volume average mixing rule uses an arithmetic mean weighted by the volume fractions,

$$X = \sum_{i=1}^{n} V_{f,i} X_i \tag{2-1}$$

where n is the number of phases, $V_{f,i}$ is the volume fraction of phase i, X is the effective material property, and X_i is the material property of phase i.

Mass Average

The mass average mixing rule uses an arithmetic mean weighted by the mass fractions,

$$X = \sum_{i=1}^{n} w_i X_i \tag{2-2}$$

where w_i is the mass fraction of phase i,

$$w_i = \frac{V_{f,i} \rho_i}{\rho} \tag{2-3}$$

Here, ρ is the effective density and ρ_i is the density of phase i.

The mass average mixing rule is not available for density.

Harmonic Volume Average

The harmonic volume average mixing rule uses a harmonic mean weighted by the volume fractions,

$$X = \left(\sum_{i=1}^{n} \frac{V_{f,i}}{X_i} \right)^{-1} \tag{2-4}$$

Harmonic Mass Average

The harmonic mass average mixing rule uses a harmonic mean weighted by the mass fractions,

$$X = \left(\sum_{i=1}^{n} \frac{w_i}{X_i} \right)^{-1} \tag{2-5}$$

The harmonic mass average mixing rule is not available for density.

Power Law

The power law mixing rule uses a power law distribution,

$$X = \prod_{i=1}^{n} X_i^{V_{f,i}} \tag{2-6}$$

Heaviside Function

The Heaviside Function mixing rule uses an arithmetic mean weighted by a smooth step function of the volume fraction:

$$X = \sum_{i=1}^{n} f_{hs,i} X_i \tag{2-7}$$

$$f_{hs,i \neq c} = \mathrm{H}\left(V_{f,i} - 0{,}5, \frac{l_{\mathrm{mix}}}{2} \right)$$

$$f_{hs,c} = 1 - \sum_{i \neq c}^{n} f_{hs,i}$$

where $f_{hs,i}$ is the fraction used for phase i, $f_{hs,c}$ is the fraction corresponding to the constrained phase, and l_{mix} is a user-defined **mixing parameter** (default: 0.8) defining the size of the transition zone between phases for the smooth Heaviside function H.

MIXING RULES FOR COMPOSITE MATERIALS

The effective structural material properties of composites depend on the material properties of the constituents and their geometric arrangements. These can be approximated either analytically or numerically. The numerical approach is applicable to a broader range of geometrical configurations, but is more complicated to use. The Effective Material includes a list of analytical approaches that are simple and handy, aimed at cases where an orthotropic fiber acts as reinforcement of an isotropic matrix. These methods are taken from Ref. 1, Ref. 2, and Ref. 3.

In this section, the properties of the fiber reinforcements are denoted by subscript f, while properties of the matrix are denoted by m. The mixing rules for composites are only available to a specific set of material properties. These rules should be used together with a Volume Averaged mixing rule for density.

Voigt–Reuss Model

The Voigt–Reuss model, also known as Rule of Mixtures, is an analytical method to compute homogenized material properties of composites. This method works well for cases where continuous orthotropic fibers are embedded in the isotropic matrix.

TABLE 2-4: EFFECTIVE PROPERTIES FOR THE VOIGT–REUSS MODEL.

MATERIAL PARAMETER	PROPERTY GROUP	COMPONENT	EFFECTIVE MIXED PROPERTY
Young's modulus, E	Orthotropic	E_{11}	$E_{f11}V_{f,f} + E_m V_{f,m}$
		E_{22}	$\dfrac{E_{f22}E_m}{E_m V_{f,f} + E_{f22}V_{f,m}}$
		E_{33}	$\dfrac{E_{f33}E_m}{E_m V_{f,f} + E_{f33}V_{f,m}}$
Shear rate, G	Orthotropic	G_{12}	$\dfrac{G_{f12}G_m}{G_m V_{f,f} + G_{f12}V_{f,m}}$
		G_{23}	$\dfrac{G_{f23}G_m}{G_m V_{f,f} + G_{f23}V_{f,m}}$
		G_{13}	$\dfrac{G_{f13}G_m}{G_m V_{f,f} + G_{f13}V_{f,m}}$

TABLE 2-4: EFFECTIVE PROPERTIES FOR THE VOIGT–REUSS MODEL.

MATERIAL PARAMETER	PROPERTY GROUP	COMPONENT	EFFECTIVE MIXED PROPERTY
Poisson's ratio, ν	Orthotropic	ν_{12}	$\nu_{f12}V_{f,f} + \nu_m V_{f,m}$
		ν_{23}	$\nu_{f23}V_{f,f} + \nu_m V_{f,m}$
		ν_{13}	$\nu_{f13}V_{f,f} + \nu_m V_{f,m}$
Coefficient of thermal expansion, α		α_{11}	$\frac{\alpha_{f11}E_{f11}V_{f,f} + \alpha_m E_m V_{f,m}}{E_{f11}V_{f,f} + E_m V_{f,m}}$
		α_{22}	$\alpha_{f22}V_{f,f} + \alpha_m V_{f,m}$
		α_{33}	$\alpha_{f33}V_{f,f} + \alpha_m V_{f,m}$
Coefficient of hygroscopic swelling, β		β_{11}	$\frac{\beta_{f11}E_{f11}V_{f,f} + \beta_m E_m V_{f,m}}{E_{f11}V_{f,f} + E_m V_{f,m}}$
		β_{22}	$\beta_{f22}V_{f,f} + \beta_m V_{f,m}$
		β_{33}	$\beta_{f33}V_{f,f} + \beta_m V_{f,m}$
Thermal conductivity, k	Basic	k_{11}	$k_{f11}V_{f,f} + k_m V_{f,m}$
		k_{22}	$\frac{k_{f22}k_m}{k_m V_{f,f} + k_{f22}V_{f,m}}$
		k_{33}	$\frac{k_{f33}k_m}{k_m V_{f,f} + k_{f33}V_{f,m}}$
Electric conductivity, σ	Basic	σ_{11}	$\sigma_{f11}V_{f,f} + \sigma_m V_{f,m}$
		σ_{22}	$\frac{\sigma_{f22}\sigma_m}{\sigma_m V_{f,f} + \sigma_{f22}V_{f,m}}$
		σ_{33}	$\frac{\sigma_{f33}\sigma_m}{\sigma_m V_{f,f} + \sigma_{f33}V_{f,m}}$

Modified Voigt–Reuss Model

The Voigt–Reuss model provides results in the longitudinal directions that are in good agreement with experiments, but not in the transverse directions. The modified Voigt–Reuss model, or modified rule of mixture, provides corrections in the transverse direction. The following correction factors are used:

$$\eta_f = \frac{E_{f11}V_{f,f} + \left(\left(1 - \nu_{f12}^2 \frac{E_{f22}}{E_{f11}}\right)E_m + \nu_m \nu_{f12} E_{f22}\right)V_{f,m}}{E_{f11}V_{f,f} + E_m V_{f,m}} \tag{2-8}$$

$$\eta_m = \frac{E_m V_{f,m} + ((1 - \nu_m^2)E_{f11} + \nu_m \nu_{f12} E_m)V_{f,f}}{E_{f11}V_{f,f} + E_m V_{f,m}} \tag{2-9}$$

$$\eta_g = 0{,}6 \tag{2-10}$$

TABLE 2-5: EFFECTIVE PROPERTIES FOR THE MODIFIED VOIGT–REUSS MODEL.

MATERIAL PARAMETER	PROPERTY GROUP	COMPONENT	EFFECTIVE MIXED PROPERTY
Young's modulus, E	Orthotropic	E_{11}	$E_{f11}V_{f,f} + E_m V_{f,m}$
		E_{22}	$\frac{E_{f22}E_m}{E_m V_{f,f}\eta_f + E_{f22}V_{f,m}\eta_m}$
		E_{33}	$\frac{E_{f33}E_m}{E_m V_{f,f}\eta_f + E_{f33}V_{f,m}\eta_m}$
Shear rate, G	Orthotropic	G_{12}	$\frac{(V_{f,f} + V_{f,m}\eta_g)G_{f12}G_m}{G_m V_{f,f} + \eta_g G_{f12}V_{f,m}}$
		G_{23}	$\frac{G_{f23}G_m}{G_m V_{f,f} + G_{f23}V_{f,m}}$
		G_{13}	$\frac{(V_{f,f} + V_{f,m}\eta_g)G_{f13}G_m}{G_m V_{f,f} + \eta_g G_{f13}V_{f,m}}$
Poisson's ratio, ν	Orthotropic	ν_{12}	$\nu_{f12}V_{f,f} + \nu_m V_{f,m}$
		ν_{23}	$\nu_{f23}V_{f,f} + \nu_m V_{f,m}$
		ν_{13}	$\nu_{f13}V_{f,f} + \nu_m V_{f,m}$

Chamis Model

The Chamis model works well for continuous orthotropic fibers embedded in the isotropic matrix. Unlike the Voigt–Reuss model, it can be used for randomly scattered unidirectional fibers.

TABLE 2-6: EFFECTIVE PROPERTIES FOR THE CHAMIS MODEL.

MATERIAL PARAMETER	PROPERTY GROUP	COMPONENT	EFFECTIVE MIXED PROPERTY
Young's modulus, E	Orthotropic	E_{11}	$E_{f11}V_{f,f} + E_m V_{f,m}$
		E_{22}	$\dfrac{E_m}{1-\sqrt{V_{f,f}}\left(1-\dfrac{E_m}{E_{f22}}\right)}$
		E_{33}	$\dfrac{E_m}{1-\sqrt{V_{f,f}}\left(1-\dfrac{E_m}{E_{f33}}\right)}$
Shear rate, G	Orthotropic	G_{12}	$\dfrac{G_m}{1-\sqrt{V_{f,f}}\left(1-\dfrac{G_m}{G_{f12}}\right)}$
		G_{23}	$\dfrac{G_m}{1-\sqrt{V_{f,f}}\left(1-\dfrac{G_m}{G_{f23}}\right)}$
		G_{13}	$\dfrac{G_m}{1-\sqrt{V_{f,f}}\left(1-\dfrac{G_m}{G_{f13}}\right)}$
Poisson's ratio, ν	Orthotropic	ν_{12}	$\nu_{f12}V_{f,f} + \nu_m V_{f,m}$
		ν_{23}	$\dfrac{E_{f22}}{2G_{f23}} - 1$
		ν_{13}	$\nu_{f13}V_{f,f} + \nu_m V_{f,m}$

TABLE 2-6: EFFECTIVE PROPERTIES FOR THE CHAMIS MODEL.

MATERIAL PARAMETER	PROPERTY GROUP	COMPONENT	EFFECTIVE MIXED PROPERTY
Coefficient of thermal expansion, α		α_{11}	$\frac{\alpha_{f11}E_{f11}V_{f,f}+\alpha_m E_m V_{f,m}}{E_{f11}V_{f,f}+E_m V_{f,m}}$
		α_{22}	$\alpha_{f22}\sqrt{V_{f,f}}+\alpha_m(1-\sqrt{V_{f,f}})\left(1+\frac{V_{f,f}\nu_m E_{f11}}{E_{f11}V_{f,f}+E_m V_{f,m}}\right)$
		α_{33}	$\alpha_{f33}\sqrt{V_{f,f}}+\alpha_m(1-\sqrt{V_{f,f}})\left(1+\frac{V_{f,f}\nu_m E_{f11}}{E_{f11}V_{f,f}+E_m V_{f,m}}\right)$
Coefficient of hygroscopic swelling, β		β_{11}	$\frac{\beta_{f11}E_{f11}V_{f,f}+\beta_m E_m V_{f,m}}{E_{f11}V_{f,f}+E_m V_{f,m}}$
		β_{22}	$\beta_{f22}\sqrt{V_{f,f}}+\beta_m(1-\sqrt{V_{f,f}})\left(1+\frac{V_{f,f}\nu_m E_{f11}}{E_{f11}V_{f,f}+E_m V_{f,m}}\right)$
		β_{33}	$\beta_{f33}\sqrt{V_{f,f}}+\beta_m(1-\sqrt{V_{f,f}})\left(1+\frac{V_{f,f}\nu_m E_{f11}}{E_{f11}V_{f,f}+E_m V_{f,m}}\right)$
Thermal conductivity, k	Basic	k_{11}	$k_{f11}V_{f,f}+k_m V_{f,m}$
		k_{22}	$k_m(1-\sqrt{V_{f,f}})+\frac{k_m}{1-\sqrt{V_{f,f}}\left(1-\frac{k_m}{k_{f22}}\right)}$
		k_{33}	$k_m(1-\sqrt{V_{f,f}})+\frac{k_m}{1-\sqrt{V_{f,f}}\left(1-\frac{k_m}{k_{f33}}\right)}$

Halpin–Tsai Model

The Halpin–Tsai model is a semi-empirical method to computed homogenized properties of composites based on self-consistent micromechanics as well as experimental curve fitting. This method works well for continuous orthotropic fibers or discontinuous orthotropic fibers embedded in the isotropic matrix, including short fibers.

The Halpin–Tsai model includes empirical factors ζ for the components of the elastic modulus and are a measure of the reinforcement that depends on fiber geometry. Some

typical values for the orthotropic Young's modulus are depicted in the table value. The value of ζ for the components of G is typically 1.

TABLE 2-7: TYPICAL VALUES OF ζ FOR YOUNG'S MODULUS E.

COMPONENT	ζ FOR DISCONTINUOUS FIBERS	ζ FOR CONTINUOUS FIBERS
E_{11}	$2\frac{l_{11}}{t_{33}}$	∞
E_{22}	$2\frac{w_{11}}{t_{33}}$	0

In the table above, l_{11} is the fiber length in the 11 direction, w_{22} is the fiber width in the 22 direction, and t_{33} is the fiber thickness in the 33 direction.

The effective material properties are given by:

TABLE 2-8: EFFECTIVE PROPERTIES FOR HALPIN–TSAI MODEL.

MATERIAL PARAMETER	PROPERTY GROUP	COMPONENT	EFFECTIVE MIXED PROPERTY
Young's modulus, E	Orthotropic	E_{11}	$E_m\frac{1+\varsigma_{E,11}\eta_{E,11}V_{f,f}}{1-\eta_{E,11}V_{f,f}}$ $\eta_{E,11}=\frac{\frac{E_{f11}}{E_m}-1}{\frac{E_{f11}}{E_m}+\varsigma_{E,11}}$
		E_{22}	$E_m\frac{1+\varsigma_{E,22}\eta_{E,22}V_{f,f}}{1-\eta_{E,22}V_{f,f}}$ $\eta_{E,22}=\frac{\frac{E_{f22}}{E_m}-1}{\frac{E_{f22}}{E_m}+\varsigma_{E,22}}$
		E_{33}	$E_m\frac{1+\varsigma_{E,33}\eta_{E,33}V_{f,f}}{1-\eta_{E,33}V_{f,f}}$ $\eta_{E,33}=\frac{\frac{E_{f33}}{E_m}-1}{\frac{E_{f33}}{E_m}+\varsigma_{E,33}}$

TABLE 2-8: EFFECTIVE PROPERTIES FOR HALPIN–TSAI MODEL.

MATERIAL PARAMETER	PROPERTY GROUP	COMPONENT	EFFECTIVE MIXED PROPERTY
Shear rate, G	Orthotropic	G_{12}	$G_m \frac{1+\varsigma_{G,12}\eta_{G,12}V_{f,f}}{1-\eta_{G,12}V_{f,f}}$ $\eta_{G,12} = \frac{\frac{G_{f12}}{G_m}-1}{\frac{G_{f12}}{G_m}+\varsigma_{G,12}}$
		G_{23}	$G_m \frac{1+\varsigma_{G,23}\eta_{G,23}V_{f,f}}{1-\eta_{G,23}V_{f,f}}$ $\eta_{G,23} = \frac{\frac{G_{f23}}{G_m}-1}{\frac{G_{f23}}{G_m}+\varsigma_{G,23}}$
		G_{13}	$G_m \frac{1+\varsigma_{G,13}\eta_{G,13}V_{f,f}}{1-\eta_{G,13}V_{f,f}}$ $\eta_{G,13} = \frac{\frac{G_{f13}}{G_m}-1}{\frac{G_{f13}}{G_m}+\varsigma_{G,13}}$
Poisson's ratio, ν	Orthotropic	ν_{12}	$\nu_{f12}V_{f,f}+\nu_m V_{f,m}$
		ν_{23}	$\nu_{f23}V_{f,f}+\nu_m V_{f,m}$
		ν_{13}	$\nu_{f13}V_{f,f}+\nu_m V_{f,m}$

Halpin–Tsai–Nielsen Model

The Halpin–Tsai–Nielsen model extends the Halpin–Tsai model by accounting for the maximum packing fraction of the reinforcement, ϕ_{max} (default 0.82). Typical values of the maximum packing factor are listed in Table 2-9.

TABLE 2-9: MAXIMUM PACKING FACTORS.

FIBER ARRANGEMENT	ϕ_{max}
Square array of fibers	0.785
Hexagonal array of fibers	0.907
Other arrangements	0.82

The Halpin–Tsai–Nielsen model introduces an auxiliary variable,

$$\Psi = 1 + \left(\frac{1 - \phi_{\max}}{\phi_{\max}^2} \right) V_{f,f} \tag{2-11}$$

and the effective properties are given by Table 2-10.

TABLE 2-10: EFFECTIVE PROPERTIES FOR THE HALPIN–TSAI–NIELSEN MODEL.

MATERIAL PARAMETER	PROPERTY GROUP	COMPONENT	EFFECTIVE MIXED PROPERTY
Young's modulus, E	Orthotropic	E_{11}	$E_m \frac{1 + \varsigma_{E,11}\eta_{E,11}V_{f,f}}{1 - \eta_{E,11}\Phi V_{f,f}}$ $\eta_{E,11} = \frac{\frac{E_{f11}}{E_m} - 1}{\frac{E_{f11}}{E_m} + \varsigma_{E,11}}$
		E_{22}	$E_m \frac{1 + \varsigma_{E,22}\eta_{E,22}V_{f,f}}{1 - \eta_{E,22}\Phi V_{f,f}}$ $\eta_{E,22} = \frac{\frac{E_{f22}}{E_m} - 1}{\frac{E_{f22}}{E_m} + \varsigma_{E,22}}$
		E_{33}	$E_m \frac{1 + \varsigma_{E,33}\eta_{E,33}V_{f,f}}{1 - \eta_{E,33}\Phi V_{f,f}}$ $\eta_{E,33} = \frac{\frac{E_{f33}}{E_m} - 1}{\frac{E_{f33}}{E_m} + \varsigma_{E,33}}$

TABLE 2-10: EFFECTIVE PROPERTIES FOR THE HALPIN–TSAI–NIELSEN MODEL.

MATERIAL PARAMETER	PROPERTY GROUP	COMPONENT	EFFECTIVE MIXED PROPERTY
Shear rate, G	Orthotropic	G_{12}	$G_m \frac{1+\varsigma_{G,12}\eta_{G,12}V_{f,f}}{1-\eta_{G,12}\Phi V_{f,f}}$ $\eta_{G,12} = \frac{\frac{G_{f12}}{G_m}-1}{\frac{G_{f12}}{G_m}+\varsigma_{G,12}}$
		G_{23}	$G_m \frac{1+\varsigma_{G,23}\eta_{G,23}V_{f,f}}{1-\eta_{G,23}\Phi V_{f,f}}$ $\eta_{G,23} = \frac{\frac{G_{f23}}{G_m}-1}{\frac{G_{f23}}{G_m}+\varsigma_{G,23}}$
		G_{13}	$G_m \frac{1+\varsigma_{G,13}\eta_{G,13}V_{f,f}}{1-\eta_{G,13}\Phi V_{f,f}}$ $\eta_{G,13} = \frac{\frac{G_{f13}}{G_m}-1}{\frac{G_{f13}}{G_m}+\varsigma_{G,13}}$
Poisson's ratio, ν	Orthotropic	ν_{12}	$\nu_{f12}V_{f,f}+\nu_m V_{f,m}$
		ν_{23}	$\nu_{f23}V_{f,f}+\nu_m V_{f,m}$
		ν_{13}	$\nu_{f13}V_{f,f}+\nu_m V_{f,m}$

Hashin–Rosen Model

The Hashin–Rosen model is based on a composite cylinder assemblage (CCA). In order to compute homogenized material properties, additional properties are computed first.

The bulk modulus of fiber, K_f, and matrix, K_m, under longitudinal strain are

$$K_{f,11} = \frac{E_{f,11}}{2(1-2\nu_{f,12})(1+\nu_{f,12})} \tag{2-12}$$

$$K_m = \frac{E_m}{2(1-2\nu_m)(1+\nu_m)} \tag{2-13}$$

The bulk modulus of the composite K under longitudinal strain is

$$K = \frac{K_m(K_{f,11}+G_m)V_{f,m}+K_{f,11}(K_m+G_m)V_{f,f}}{(K_{f,11}+G_m)V_{f,m}+(K_m+G_m)V_{f,f}} \tag{2-14}$$

In this model, the effective shear modulus of the composite G_{23} is approximated from the quadratic equation

$$A\left(\frac{G_{23}}{G_m}\right)^2 + 2B\left(\frac{G_{23}}{G_m}\right) + C = 0 \tag{2-15}$$

where

$$A = 3V_{f,f}V_{f,m}\left(\frac{G_{23}}{G_m}-1\right)\left(\frac{G_{23}}{G_m}+\eta_f\right) \tag{2-16}$$

$$+\left(\frac{G_{23}}{G_m}\eta_m+\eta_f\eta_m-\left(\frac{G_{23}}{G_m}\eta_m-\eta_f\right)V_{f,f}^3\right)\left(\left(\frac{G_{23}}{G_m}-1\right)V_{f,f}\eta_m-\left(\frac{G_{23}}{G_m}+1\right)\right)$$

$$B = -3V_{f,f}V_{f,m}\left(\frac{G_{23}}{G_m}-1\right)\left(\frac{G_{23}}{G_m}+\eta_f\right) \tag{2-17}$$

$$+\frac{V_{f,f}}{2}(\eta_m+1)\left(\frac{G_{23}}{G_m}-1\right)\left(\frac{G_{23}}{G_m}+\eta_f-\left(\frac{G_{23}}{G_m}\eta_m-\eta_f\right)V_{f,f}^3\right)$$

$$+\frac{1}{2}\left(\frac{G_{23}}{G_m}\eta_m-\left(\frac{G_{23}}{G_m}-1\right)V_{f,f}+1\right)\left((\eta_f+1)\left(\frac{G_{23}}{G_m}+\eta_f\right)-2\left(\frac{G_{23}}{G_m}\eta_m-\eta_f\right)V_{f,f}^3\right)$$

$$C = -3V_{f,f}V_{f,m}\left(\frac{G_{23}}{G_m}-1\right)\left(\frac{G_{23}}{G_m}+\eta_f\right) \tag{2-18}$$

$$+\left(\frac{G_{23}}{G_m}\eta_m+\left(\frac{G_{23}}{G_m}-1\right)V_{f,f}+1\right)\left(\frac{G_{23}}{G_m}+\eta_f+\left(\frac{G_{23}}{G_m}\eta_m-\eta_f\right)V_{f,f}^3\right)$$

and

$$\eta_m = 3-\nu_m$$

$$\eta_f = 3 - \nu_{f,23}$$

The effective material properties are then given by Table 2-11.

TABLE 2-11: EFFECTIVE PROPERTIES FOR HASHIN–ROSEN MODEL.

MATERIAL PARAMETER	PROPERTY GROUP	COMPONENT	EFFECTIVE MIXED PROPERTY
Young's modulus, E	Orthotropic	E_{11}	$E_{f11}V_{f,f} + E_m V_{f,m} + \frac{4V_{f,f}V_{f,m}(\nu_{f,12} - \nu_m)^2}{\frac{V_{f,f}}{K_m} + \frac{1}{G_m} + \frac{V_{f,m}}{K_f}}$
		E_{22}	$2(1+\nu_{f,23})G_{23}$
		E_{33}	$2(1+\nu_{f,23})G_{23}$
Shear rate, G	Orthotropic	G_{12}	$G_m \frac{(1+V_{f,f})G_{f12} + G_m V_{f,m}}{G_{f12}V_{f,m} + (1+V_{f,f})G_m}$
		G_{23}	Positive value of $G_m \frac{-2B \mp \sqrt{4B^2 - 4Ac}}{2A}$
		G_{13}	$G_m \frac{(1+V_{f,f})G_{f13} + G_m V_{f,m}}{G_{f13}V_{f,m} + (1+V_{f,f})G_m}$

TABLE 2-11: EFFECTIVE PROPERTIES FOR HASHIN–ROSEN MODEL.

MATERIAL PARAMETER	PROPERTY GROUP	COMPONENT	EFFECTIVE MIXED PROPERTY
Poisson's ratio, ν	Orthotropic	ν_{12}	$\nu_{f12}V_{f,f}+\nu_m V_{f,m} + \dfrac{V_{f,f}V_{f,m}(\nu_{f,12}-\nu_m)\left(\frac{1}{K_m}-\frac{1}{K_f}\right)}{\frac{V_{f,f}}{K_m}+\frac{1}{G_m}+\frac{V_{f,m}}{K_f}}$
		ν_{23}	$\dfrac{K-mG_{23}}{K+mG_{23}}$ $m = 1+4K\dfrac{\nu_{12}^2}{E_{11}}$
		ν_{13}	$\nu_{f13}V_{f,f}+\nu_m V_{f,m} + \dfrac{V_{f,f}V_{f,m}(\nu_{f,13}-\nu_m)\left(\frac{1}{K_m}-\frac{1}{K_f}\right)}{\frac{V_{f,f}}{K_m}+\frac{1}{G_m}+\frac{V_{f,m}}{K_f}}$

OTHER MIXING RULES IN THE MULTIPHASE MATERIAL

Krieger-Type Viscosity

The effective dynamic viscosity μ in suspensions may be modeled using the Krieger-type model as

$$\mu = \mu_1\left(1-\frac{V_{f,2}}{V_{f,\max}}\right)^{-2,5V_{f,max}} \tag{2-19}$$

When **Krieger type** viscosity is selected, enter a value or expression for the **Maximum packing concentration** $V_{f,\max}$ (dimensionless). The default is 0.62.

The Krieger type mixing rule is available only for the dynamic viscosity parameter in the basic property group.

REFERENCES FOR MIXING RULES

1. K.K. Chawala, "Composite Materials: Science and Engineering," Springer, 2012.

2. J. Aboudi, S.M. Arnold, and B.A. Bednarcyk, "Micromechanics of Composite Materials: A Generalized Multiscale Analysis Approach," Elsevier, 2013.

3. J.C. Halpin and J.L. Kardos, "The Halpin–Tsai Equations: A Review," *Polymer Eng. Science.*, vol. 16, no. 5, pp. 344–352, 1976.

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 2-13 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 2-12.

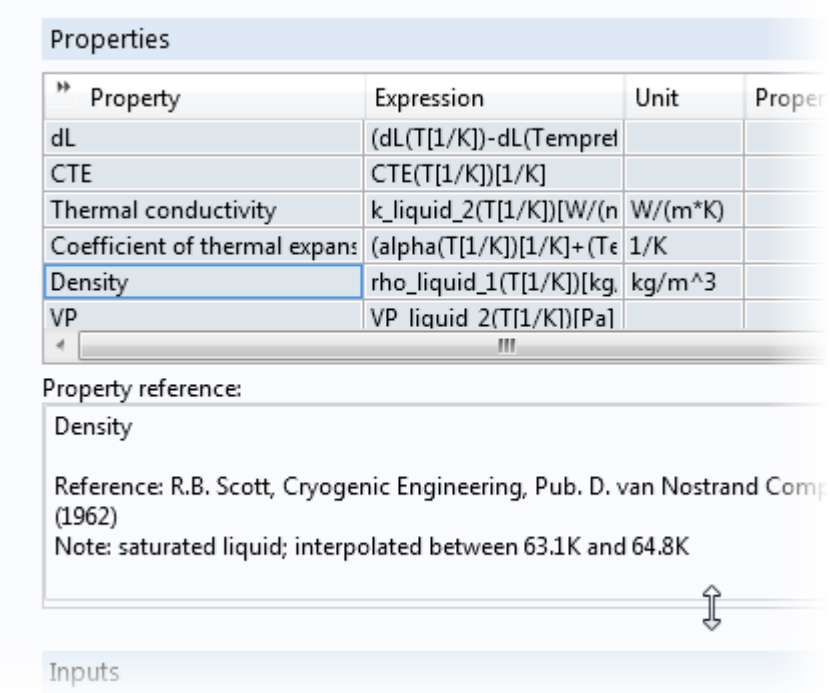


Figure 2-12: An example of where you can find the property reference information for a material. In this example, Density has this information available in the Property reference section. You can hover over the section and drag to expand it if required.

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 2-12. 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 2-12: 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 2-13: 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
Electric 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 2-13: 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 2-13.

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.

ELECTRIC 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 default model inputs. See Default Model Inputs in the *COMSOL Multiphysics Reference Manual*.

All physical quantities that can act as model inputs declare and define common variables that are 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** node's 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 2-14: 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
Electric 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 2-14: 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
- Shell Geometric Properties
- Magnetostrictive Models
- Optical Models
- Piezoelectric Models
- Piezoresistive Models
- Semiconductor Material Properties
- 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
- Thermal Expansion Material Properties
- External Material Properties
- Fluid Flow Material Properties: Inelastic Non-Newtonian Material Model

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 are used by the Acoustics Module.

TABLE 2-15: ACOUSTICS MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
ANISOTROPIC ACOUSTICS MODEL		
Effective density	rho_eff_iso (3-by-3 matrix)	m^2/s
Effective bulk modulus	K_eff	Pa
ATTENUATION AND DISSIPATION MODEL		
Sound diffusivity	delta_diff	m^2/s
NONLINEAR MODEL		
Parameter of nonlinearity	BA	1
POROACOUSTICS MODEL		

TABLE 2-15: ACOUSTICS MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Flow resistivity	Rf	$Pa·s/m^2$
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)
VISCOUS MODEL		
Bulk viscosity	muB	Pa·s

Electric Discharge Material Properties

The following material property groups are used for electric discharge modeling in the Electric Discharge Module.

TABLE 2-16: ELECTRIC DISCHARGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
CHARGE TRANSPORT IN GASES (CTG)		
Electron mobility	mu_e_iso (3-by-3 matrix)	$m^2/(V·s)$
Positive ion mobility	mu_p_iso (3-by-3 matrix)	$m^2/(V·s)$
Negative ion mobility	mu_n_iso (3-by-3 matrix)	$m^2/(V·s)$
Ionization coefficient	alpha	1/m
Attachment coefficient	eta	1/m
Electron–ion recombination coefficient	beta_ep	m^3/s
Ion–Ion recombination coefficient	beta_pn	m^3/s
CHARGE TRANSPORT IN LIQUIDS (CTL)		
Electron mobility	mu_e_iso (3-by-3 matrix)	$m^2/(V·s)$

TABLE 2-16: ELECTRIC DISCHARGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Positive ion mobility	mu_p_iso (3-by-3 matrix)	$m^2/(V \cdot s)$
Negative ion mobility	mu_n_iso (3-by-3 matrix)	$m^2/(V \cdot s)$
Electron–ion recombination coefficient	beta_ep	m^3/s
Ion–Ion recombination coefficient	beta_pn	m^3/s
Attachment time constant	tau_a	s
Number density of ionizable species	n_ioni	$1/m^3$
Molecular separation distance	a	m
Effective electron mass	m_eff	kg
Ionization potential	phi_Delta	V
Parameter	phi_gamma	V
CHARGE TRANSPORT IN SOLIDS (CTS)		
Electron band mobility	mu_be_iso (3-by-3 matrix)	$m^2/(V \cdot s)$
Hole band mobility	mu_bh_iso (3-by-3 matrix)	$m^2/(V \cdot s)$
Depth of shallow traps, electrons	phi_st_e	V
Depth of shallow traps, holes	phi_st_h	V
Attempt-to-escape frequency for trapped electrons	nu_te	1/s
Attempt-to-escape frequency for trapped holes	nu_th	1/s
Detrapping barrier height for trapped electrons	phi_te	V
Detrapping barrier height for trapped holes	phi_th	V
Trapping rate for electrons	B_e	1/s
Trapping rate for holes	B_h	1/s
Number density of deep traps for electrons	n_0_te	$1/m^3$
Number density of deep traps for holes	n_0_th	$1/m^3$
Trapped electron-trapped hole recombination coefficient	C_0	m^3/s
Electron-trapped hole recombination coefficient	C_1	m^3/s
Trapped electron–hole recombination coefficient	C_2	m^3/s
Electron–hole recombination coefficient	C_3	m^3/s

Electrochemistry Material Properties

The following material property groups are used for electrochemistry in the Battery Design Module, Corrosion Module, Electrochemistry Module, Electrodeposition Module, and Fuel Cell & Electrolyzer Module.

TABLE 2-17: ELECTROCHEMISTRY MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
DIFFUSION TIME CONSTANT		
Diffusion time constant	taumat	s
DIMENSIONLESS CHARGE EXCHANGE CURRENT		
Dimensionless charge exchange current	J0mat	1
ELECTROLYTE CONDUCTIVITY		
Electrolyte conductivity	sigmal	S/m
ELECTROLYTE SALT CONCENTRATION		
Electrolyte salt concentration	cElsalt	mol/m^3
EQUILIBRIUM CONCENTRATION	This node defines the equilibrium concentration for a given potential in intercalating electrode materials.	
Equilibrium concentration	cEeqref	mol/m^3
EQUILIBRIUM DEGREE OF CONVERSION	This node defines the equilibrium degree of conversion for a given potential in battery electrode materials.	
Equilibrium degree of conversion	docEq	1
EQUILIBRIUM POTENTIAL		
Equilibrium potential	Eeq	V
Reference concentration	cEeqref	mol/m^3
Temperature derivative of equilibrium potential	dEeqdT	V/K
EQUILIBRIUM POTENTIAL (USING DEGREES OF CONVERSION AS MODEL INPUT)	This node defines the equilibrium potential for a given degree of conversion in battery electrodes.	
Equilibrium potential	Eeq	V
Temperature derivative of equilibrium potential	dEeqdT	V/K

TABLE 2-17: ELECTROCHEMISTRY MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
INTERCALATION STRAIN	This node defines a material property for the volumetric strain to be used with the **Linear Elastic Material** > **Intercalation Strain** node in Solid Mechanics.	
LINEARIZED RESISTIVITY	This 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
LOCAL CURRENT DENSITY		
Local current density expression	ilocmat	A/m^2
LUMPED BATTERY, BASIC	This node contains a set of lumped battery parameters for the Battery Pack interface.	
Initial battery cell capacity	Q_cellmat	C
Ohmic overpotential at 1C	eta_ir1Cmat	V
Open circuit voltage at reference temperature	Eocvmat	V
Reference temperature	Trefmat	K
Temperature derivative of open circuit voltage	dEocvdTmat	V/K
MEMBRANE CROSSOVER	This node defines the permeation coefficients of various gases in a membrane.	
Permeation coefficient	Psi_H2	s·mol/kg
Permeation coefficient	Psi_O2	s·mol/kg
Permeation coefficient	Psi_N2	s·mol/kg
OPERATIONAL ELECTRODE STATE-OF-CHARGE		
Maximum electrode state-of-charge	socmax	1
Minimum electrode state-of-charge	socmin	1
POLYMER ELECTROLYTE WATER ABSORPTION-DESORPTION	This node defines the water absorption-desorption rate constant at the interface between a polymer electrolyte and an adjacent gas.	

TABLE 2-17: ELECTROCHEMISTRY MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Absorption-desorption rate constant	k_abs_dsp	mol/(m^2·s)
POLYMER ELECTROLYTE WATER TRANSPORT	This node defines the transport properties of water in a polymer electrolyte.	
Water transport coefficient	alpha	s·mol^2/(kg·m^3)
Electroosmotic coefficient	xi	1
RESISTOR–CAPACITOR PAIR		
RC time constant	tau_RCmat	s
RC potential at 1 C	E_RC1Cmat	V
SPECIES PROPERTIES		
Transport number	transNum	1
WATER VAPOR-LIQUID PHASE TRANSITION	This node defines the heat of vaporization and vapor pressure of an aqueous electrolyte.	
Vapor pressure	p_vap	Pa
Heat of vaporization	H_vap	J/mol

Electromagnetic Models

These following material property groups are used for various electromagnetic material models in the AC/DC Module, RF Module, and Wave Optics Module.

TABLE 2-18: ELECTROMAGNETIC MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
B-H CURVE	This node is only available with the AC/DC Module.	
Magnetic flux density norm	normB	T
Magnetic field norm	normH	A/m
Magnetic coenergy density	Wpm	J/m^3
B-HS CURVE		
Magnetic flux density norm	normB	T
Shifted magnetic field norm	normHs	A/m
Coercive magnetic field	Hc	A/m
Magnetic coenergy density	Wpm	J/m^3
COERCIVE MAGNETIC FIELD		

TABLE 2-18: ELECTROMAGNETIC MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Coercive magnetic field	Hc	A/m
DIELECTRIC LOSSES		
Dielectric loss factor	eta_epsilon	1
Relative permittivity (imaginary part)	epsilonBis	1
Relative permittivity (real part)	epsilonPrim	1
E-J CHARACTERISTIC		
Electric field norm	normE	V/M
EFFECTIVE B-H CURVE	This node is only available with the AC/DC Module.	
Effective magnetic flux density norm	normBeff	T
Effective magnetic field norm	normHeff	A/m
EFFECTIVE H-B CURVE		
Effective magnetic field norm	normHeff	A/m
FERROELECTRIC		
Saturation polarization	Psat	C/m^2
Initial electric susceptibility	chie0_iso (3-by-3 matrix)	1
Saturation electrostriction	lambdase	1
Pinning loss	kJAe_iso (3-by-3 matrix)	V/m
Polarization reversibility	cJAe_iso (3-by-3 matrix)	1
Interdomain coupling	alphaJAe_iso (3-by-3 matrix)	m/F
Domain wall density	aJAe_iso (3-by-3 matrix)	V/m
Electrostrictive coupling matrix	Qe_iso (6-by-6 matrix)	m^4/C^2
Electrostrictive coupling matrix, Voigt notation	QVe_iso (6-by-6 matrix)	m^4/C^2
H-B CURVE		
Magnetic field norm	normH	A/m
JILES–ATHERTON MODEL PARAMETERS		
Maximum magnetization parameter	MsJA (3-by-3 matrix)	A/m

TABLE 2-18: ELECTROMAGNETIC MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Langevin slope parameter	aJA (3-by-3 matrix)	A/m
Pinning parameter	kJA (3-by-3 matrix)	A/m
Reversibility parameter	cJA (3-by-3 matrix)	1
Interdomain coupling parameter	alphaJA (3-by-3 matrix)	1
LINEARIZED RESISTIVITY	This 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 node assumes zero conductivity.	
Loss tangent, loss angle	delta	rad
Relative permittivity (real part)	epsilonPrim	1
LOSS TANGENT, DISSIPATION FACTOR	This node assumes zero conductivity.	
Loss tangent, dissipation factor	tanDelta	1
Relative permittivity (real part)	epsilonPrim	1
MAGNETIC LOSSES		
Relative permeability (imaginary part)	murBis	1
Relative permeability (real part)	murPrim	1
PERMANENT MAGNET SHIFTED B-H CURVE		
Magnetic flux density norm	normBpm	T
PERMANENT MAGNET SHIFTED H-B CURVE		
Shifted magnetic field norm	normHs	A/m
REFRACTIVE INDEX	This node assumes a relative permeability of unity and zero conductivity.	
Refractive index, imaginary part	ki	1
Refractive index	n	1
REMANENT FLUX DENSITY		
Recoil permeability	murec	1
Remanent flux density norm	normBr	T
DISPERSION MODELS > MULTIPOLE DEBYE		
Reference temperature	Tref	K

TABLE 2-18: ELECTROMAGNETIC MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Relative permittivity contributions	Gvm	1
Relaxation time	tauvm	s

Equilibrium Discharge

These material property groups are used for all the material models for Equilibrium Discharge. The material data is from Ref. 1. These property groups are used by the Plasma Module.

TABLE 2-19: EQUILIBRIUM DISCHARGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

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 is used for an ideal gas.

TABLE 2-20: GAS MATERIAL PROPERTIES.

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)

Shell Geometric Properties

The **Shell** material property group is used in connection with layered materials (it is, for example, added when you add a **Material** node by choosing **Single Layer Material** from the **Layers** submenu on the **Materials** node's context menu). This property group

contains geometric properties for the definition of a layer in the **Layer Definition** section.

TABLE 2-21: SHELL PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
SHELL		
Thickness	lth	m
Rotation	lrot	deg
Mesh elements	lne	1

Magnetostrictive Models

These material property groups are used for various magnetostrictive material models in the AC/DC Module.

TABLE 2-22: MAGNETOSTRICTIVE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
MAGNETOSTRICTIVE		
Saturation magnetization	Ms	A/m
Initial magnetic susceptibility	chi	1
Saturation magnetostriction	lambdas	1
Magnetostriction constants	lambda100	1
Magnetostriction constants	lambda111	1
STRAIN–MAGNETIZATION FORM	This material node is only available with the AC/DC Module.	
Compliance matrix	sH (6-by-6 matrix)	1/Pa
Loss factor for compliance matrix sH	eta_sH (6-by-6 matrix)	1
Piezomagnetic coupling matrix	dHT (3-by-6 matrix)	m/A
Relative permeability	murT (3-by-3 matrix)	1
STRESS–MAGNETIZATION FORM	This material node is only available with the AC/DC Module.	
Elasticity matrix	cH (6-by-6 matrix)	Pa
Loss factor for elasticity matrix cH	eta_cH (6-by-6 matrix)	1
Piezomagnetic coupling matrix	eHS (3-by-6 matrix)	T
Relative permeability	murS (3-by-3 matrix)	1

Optical Models

These material property groups appear in optical materials and are used by the Ray Optics Module and Wave Optics Module.

TABLE 2-23: OPTICAL MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
ATTENUATION COEFFICIENT		
Attenuation coefficient	alpha1	1/m
CAUCHY		
Cauchy dispersion coefficients	{ODcca1, ODcca2, ODcca3, ODcca4, ODcca5, ODcca6}	1
Reference temperature	Trefcca	K
Reference pressure	Prefcca	Pa
CONRADY		
Conrady dispersion coefficients	{ODcra1, ODcra2, ODcra3}	1
Reference temperature	Trefcra	K
Reference pressure	Prefcra	Pa
HERZBERGER		
Herzberger dispersion coefficients	{ODhza1, ODhza2, ODhza3, ODhza4, ODhza5}	1
Reference temperature	Trefhza	K
Reference pressure	Prefhza	Pa
INTERNAL TRANSMITTANCE, 2 MM SAMPLE THICKNESS		
Internal transmittance, 2 mm sample thickness	taui2	1
INTERNAL TRANSMITTANCE, 5 MM SAMPLE THICKNESS		
Internal transmittance, 5 mm sample thickness	taui5	1
INTERNAL TRANSMITTANCE, 10 MM SAMPLE THICKNESS		
Internal transmittance, 10 mm sample thickness	taui10	1
INTERNAL TRANSMITTANCE, 25 MM SAMPLE THICKNESS		

TABLE 2-23: OPTICAL MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Internal transmittance, 25 mm sample thickness	taui25	I
SCHOTT		
Schott dispersion coefficients	{ODsca1, ODsca2, ODsca3, ODsca4, ODsca5, ODsca6}	I
Reference temperature	Trefsca	K
Reference pressure	Prefsca	Pa
SCHOTT EXTENDED		
Schott extended dispersion coefficients	{ODscb1, ODscb2, ODscb3, ODscb4, ODscb5, ODscb6, ODscb7, ODscb8, ODscb9}	I
Reference temperature	Trefscb	K
Reference pressure	Prefscb	Pa
SCHOTT THERMO-OPTIC		
Schott thermo-optic dispersion coefficients	{TOsco1, TOsco2, TOsco3, TOsco4, TOsco5, TOsco6}	I
Reference temperature	Trefsco	K
SELLMEIER		
Sellmeier dispersion coefficients	{ODsma1, ODsma2, ODsma3, ODsma4, ODsma5, ODsma6}	I
Reference temperature	Trefsma	K
Reference pressure	Prefsma	Pa
SELLMEIER MODIFIED, TYPE 1		
Sellmeier modified, type 1 dispersion coefficients	{ODsmb1, ODsmb2, ODsmb3, ODsmb4, ODsmb5, ODsmb6}	I
Reference temperature	Trefsmb	K
Reference pressure	Prefsmb	Pa
SELLMEIER MODIFIED, TYPE 2		
Sellmeier modified, type 2 dispersion coefficients	{ODsmc1, ODsmc2, ODsmc3, ODsmc4, ODsmc5, ODsmc6}	I
Reference temperature	Trefsmc	K
Reference pressure	Prefsmc	Pa

TABLE 2-23: OPTICAL MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
TEMPERATURE-DEPENDENT SELLMEIER		
Temperature-dependent Sellmeier dispersion coefficients	{ODtds1, ODtds2, , ODtds30}	1

Piezoelectric Models

These material property groups for piezoelectric materials are used by the Acoustics Module, MEMS Module, and Structural Mechanics Module.

TABLE 2-24: PIEZOELECTRIC MATERIAL PROPERTY GROUPS AND PROPERTIES.

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 electric 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 electric permittivity	εS	1
Relative permittivity	epsilonrS	1

Piezoresistive Models

These material property groups for piezoresistive materials are used by the MEMS Module.

TABLE 2-25: PIEZORESISTIVE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
ELASTORESISTANCE FORM		
Elastoresistive coupling matrix	ml	Ω·m
PIEZORESISTANCE FORM		
Piezoresistive coupling matrix	Pil	A/m^2

Pyroelectric

This material property group for pyroelectric materials is used by the MEMS Module.

TABLE 2-26: PYROELECTRIC MATERIAL PROPERTY GROUP AND PROPERTY.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
PYROELECTRIC		
Total pyroelectric coefficient	{pET1, pET2, pET3}	$C/(m^2 \cdot K)$

Semiconductor Material Properties

These material property groups for all the material models in semiconductors are used by 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 2-27 are specifically for silicon in the *COMSOL Multiphysics Reference Manual*.

TABLE 2-27: SEMICONDUCTOR MATERIAL PROPERTY GROUPS AND 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

TABLE 2-27: SEMICONDUCTOR MATERIAL PROPERTY GROUPS AND 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
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
Jain–Roulston coefficient (p-type), A	Ap_jr	V	$3.5 \cdot 10^{-8}$ 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	$1/m^3$	$1.3 \cdot 10^{17}$ $1/cm^3$	Ref. 11
Conduction band fraction	alpha_sb	1	0.5	Ref. 11
GENERATION–RECOMBINATION > AUGER RECOMBINATION				
Auger recombination factor, electrons	Cn	m^6/s	$2.8 \cdot 10^{-31}$ cm^6/s (valid at 300 K)	Ref. 2

TABLE 2-27: SEMICONDUCTOR MATERIAL PROPERTY GROUPS AND 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
Auger recombination factor, holes	Cp	m^6/s	$9.9 \cdot 10^{-32}$ 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
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

TABLE 2-27: SEMICONDUCTOR MATERIAL PROPERTY GROUPS AND 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 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
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

TABLE 2-27: SEMICONDUCTOR MATERIAL PROPERTY GROUPS AND 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
Reference temperature	Tref_ct	K	300 K	Ref. 6
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	$m^2/(V\cdot s)$	$4.75 \times 10^7 cm^2/(V\cdot s)$	Ref. 8
Electron mobility reference	mun2_ls	$m^2/(V\cdot s)$	1.74×10^5 $cm^2/(V\cdot s)$	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	$m^2/(V\cdot s)$	9.93×10^7 $cm^2/(V\cdot s)$	Ref. 8
Hole mobility reference	mup2_ls	$m^2/(V\cdot s)$	8.84×10^5 $cm^2/(V\cdot s)$	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	$m^2/(V\cdot s)$	1448 $cm^2/(V\cdot s)$	Ref. 5

TABLE 2-27: SEMICONDUCTOR MATERIAL PROPERTY GROUPS AND 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	$m^2/(V·s)$	473 $cm^2/(V·s)$	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
MOBILITY MODELS > KLAASSEN UNIFIED MOBILITY MODEL				
Reference temperature	T_ref_kl	K	300 K	Ref. 13
Klaassen max electron mobility	mu_e_max_kl	$m^2/(V·s)$	1414.0 $cm^2/(V·s)$	Ref. 13
Klaassen max hole mobility	mu_h_max_kl	$m^2/(V·s)$	470.5 $cm^2/(V·s)$	Ref. 13
Klaassen min electron mobility	mu_e_min_kl	$m^2/(V·s)$	68.5 $cm^2/(V·s)$	Ref. 13
Klaassen min hole mobility	mu_h_min_kl	$m^2/(V·s)$	44.9 $cm^2/(V·s)$	Ref. 13
Klaassen lattice mobility electron exponent	theta_e_kl	1	2.285	Ref. 13
Klaassen lattice mobility hole exponent	theta_h_kl	1	2.247	Ref. 13
Klaassen dopant and carrier mobility electron exponent	alpha_e_1_kl	1	0.711	Ref. 13
Klaassen dopant and carrier mobility hole exponent	alpha_h_1_kl	1	0.719	Ref. 13
Klaassen electron reference density	N_ref_e_1_kl	$1/m^3$	9.20e16 $1/cm^3$	Ref. 13
Klaassen hole reference density	N_ref_h_1_kl	$1/m^3$	2.23e17 $1/cm^3$	Ref. 13

TABLE 2-27: SEMICONDUCTOR MATERIAL PROPERTY GROUPS AND 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
Klaassen cluster function donor coefficient	c_D_kl	1	0.21	Ref. 13
Klaassen cluster function acceptor coefficient	c_A_kl	1	0.50	Ref. 13
Klaassen cluster function donor reference density	N_ref_D_kl	$1/m^3$	4.0e20 $1/cm^3$	Ref. 13
Klaassen cluster function acceptor reference density	N_ref_A_kl	$1/m^3$	7.2e20 $1/cm^3$	Ref. 13
Klaassen P parameter BH weight	f_BH_kl	1	3.828	Ref. 13
Klaassen P parameter CW weight	f_CW_kl	1	2.459	Ref. 13
Klaassen P parameter BH prefactor	N_BH_kl	$1/m^3$	1.36e20 $1/cm^3$	Ref. 13
Klaassen P parameter CW prefactor	P_CW_kl	1	3.97e13	Ref. 13
Klaassen 1st s parameter	s_1_kl	1	0.89233	Ref. 13
Klaassen 2nd s parameter	s_2_kl	1	0.41372	Ref. 13
Klaassen 3rd s parameter	s_3_kl	1	0.19778	Ref. 13
Klaassen 4th s parameter	s_4_kl	1	0.28227	Ref. 13
Klaassen 5th s parameter	s_5_kl	1	0.005978	Ref. 13
Klaassen 6th s parameter	s_6_kl	1	1.80618	Ref. 13
Klaassen 7th s parameter	s_7_kl	1	0.72169	Ref. 13

TABLE 2-27: SEMICONDUCTOR MATERIAL PROPERTY GROUPS AND 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
Klaassen 1st r parameter	r_1_kl	1	0.7643	Ref. 13
Klaassen 2nd r parameter	r_2_kl	1	2.2999	Ref. 13
Klaassen 3rd r parameter	r_3_kl	1	6.5502	Ref. 13
Klaassen 4th r parameter	r_4_kl	1	2.3670	Ref. 13
Klaassen 5th r parameter	r_5_kl	1	-0.01552	Ref. 13
Klaassen 6th r parameter	r_6_kl	1	0.6478	Ref. 13
Klaassen electron mobility mass	m_e_kl	kg	me_const	Ref. 13
Klaassen hole mobility mass	m_h_kl	kg	1.258*me_const	Ref. 13
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
SEMICONDUCTOR MATERIAL, DENSITY-GRADIENT				
Electron effective mass, density-gradient	meDG_iso (3-by-3 matrix)	kg	N/A	

TABLE 2-27: SEMICONDUCTOR MATERIAL PROPERTY GROUPS AND 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 effective mass, density-gradient	mhDG_iso (3-by-3 matrix)	kg	N/A	
Electron effective mass, density-gradient, for penetration depth in barrier	meOx_iso (3-by-3 matrix)	kg	N/A	
Hole effective mass, density-gradient, for penetration depth in barrier	mhOG_iso (3-by-3 matrix)	kg	N/A	
Electron effective mass, density-gradient, for flux into barrier	meOxStar_iso (3-by-3 matrix)	kg	N/A	
Hole effective mass, density-gradient, for flux into barrier	mhOxStar_iso (3-by-3 matrix)	kg	N/A	

Solid Mechanics Material Properties

These material property groups appear for material models in solid mechanics. Most of these properties are used by 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 2-28: SOLID MECHANICS MATERIAL PROPERTY GROUPS AND PROPERTIES.

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
ANISOTROPIC, VOIGT NOTATION		
Elasticity matrix, Voigt notation	DV0	Pa
Loss factor for elasticity matrix D, Voigt notation	eta_DVo	1

TABLE 2-28: SOLID MECHANICS MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
BULK MODULUS AND SHEAR MODULUS		
Bulk modulus	K	N/m^2
Shear modulus	G	N/m^2
COMPLIANCE		
Compliance matrix	Dinv_iso (6-by-6 matrix)	1/Pa
Loss factor for compliance matrix	eta_Dinv_iso (6-by-6 matrix)	1
COMPLIANCE, VOIGT NOTATION		
Compliance matrix, Voigt notation	DinvVo_iso (6-by-6 matrix)	1/Pa
Loss factor for compliance matrix, Voigt notation	eta_DinvVo_iso (6-by-6 matrix)	1
CUBIC		
Elasticity constant C11	c11	Pa
Elasticity constant C12	c12	Pa
Elasticity constant C44	c44	Pa
CUBIC (3 CONSTANTS)		
Elasticity constant matrix components, Voigt notation	{c3cub1, c3cub2, c3cub3}	Pa
EFFECTIVE FLEXIBILITY MATRICES		
Extensional flexibility matrix	Da_iso (3-by-3 matrix)	s^2/kg
Bending-extensional flexibility matrix	Db_iso (3-by-3 matrix)	$s^2/(kg \cdot m)$
Bending flexibility matrix	Dd_iso (3-by-3 matrix)	$s^2/(kg \cdot m^2)$
Shear flexibility matrix	Das_iso (2-by-2 matrix)	s^2/kg
EFFECTIVE STIFFNESS MATRICES		
Extensional stiffness matrix	DA_iso (3-by-3 matrix)	N/m
Bending-extensional stiffness matrix	DB_iso (3-by-3 matrix)	N

TABLE 2-28: SOLID MECHANICS MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Bending stiffness matrix	DD_iso (3-by-3 matrix)	N·m
Shear stiffness matrix	DAs_iso (2-by-2 matrix)	N/m
Translational inertia	I0	kg/m^2
Rotational-translational inertia matrix	I1_iso (3-by-3 matrix)	kg/m
Rotational inertia matrix	I2_iso (3-by-3 matrix)	kg
Loss factor for stiffness matrix DA	eta_DA_iso (3-by-3 matrix)	1
Loss factor for stiffness matrix DB	eta_DB_iso (3-by-3 matrix)	1
Loss factor for stiffness matrix DD	eta_DD_iso (3-by-3 matrix)	1
Loss factor for stiffness matrix DAs	eta_DA_iso (2-by-2 matrix)	1
HEXAGONAL (5 CONSTANTS)		
Elasticity matrix components, Voigt notation	{c5hex1, c5hex2, c5hex3, c5hex4, c5hex5}	Pa
GENERAL STRESS–STRAIN RELATION		
Second Piola–Kirchhoff stress	S_iso (3-by-3 matrix)	N/m^2
LAMÉ PARAMETERS		
Lamé parameter λ	lambLame	N/m^2
Lamé parameter μ	muLame	N/m^2
ORTHORHOMBIC (9 CONSTANTS)		
Elasticity matrix components, Voigt notation	{c9ort1, …, c9ort9}	Pa
ORTHOTROPIC		
Young's modulus	Evector	Pa
Poisson's ratio	nuvector	1
Shear modulus	Gvector	N/m^2

TABLE 2-28: SOLID MECHANICS MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
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
TETRAGONAL (6 CONSTANTS)		
Elasticity matrix components, Voigt notation	{c6tet1, …, c6tet6}	Pa
TETRAGONAL (7 CONSTANTS)		
Elasticity matrix components, Voigt notation	{c7tet1, …, c7tet7}	Pa
TRIGONAL (6 CONSTANTS)		
Elasticity matrix components, Voigt notation	{c6tri1, …, c6tri6}	Pa
TRIGONAL (7 CONSTANTS)		
Elasticity matrix components, Voigt notation	{c7tri1, …, c7tri7}	Pa
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
Permeability	kappa	m^2
SAFETY		

TABLE 2-28: SOLID MECHANICS MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
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
TRANSVERSELY ISOTROPIC		
Young's modulus	Evect	Pa
Poisson's ratio	nuvect	Pa
Shear modulus	Gvect	Pa
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	$m^2 \cdot s^4/kg^2$
WIRE MATERIAL		
ELASTIC WIRE		
Axial stiffness	k_A	N
Mass per reference length	F_f	kg/m

- The *Structural Mechanics Module User's Guide* and Table 2-31
- The *Structural Mechanics Module User's Guide* and Table 2-29
- The *Fatigue Module User's Guide* and Table 2-30

Solid Mechanics Material Properties: Nonlinear Structural Materials Module

These material property groups for material models in solid mechanics are used by the Nonlinear Structural Materials Module.

TABLE 2-29: HYPERELASTIC, ELASTOPLASTIC, PHASE TRANSFORMATION, AND DAMAGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
ELASTOPLASTIC MATERIAL		
Hardening function	sigmagh	Pa
Hill's coefficients	{Hillcoefficients1, …, Hillcoefficients6}	$(m^2 \cdot s^4)/kg^2$
Hill's coefficients, dimensionless	{Hillcoefficients Dimensionless1, …, HillcoefficientsDimensionless6}	1
Initial tensile and shear yield stresses	{ys1, ys2, ys3, ys4, ys5, ys6}	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
CAP AND CUTOFF		
Initial pressure limit	pc0	Pa
Initial ellipse centroid	pcc0	Pa
Hardening function	pch	Pa
Hardening modulus	Kc	Pa
Maximum volumetric plastic strain	epvolmax	1
Mean stress limit	svolt	Pa
CHABOCHE		
Kinematic hardening modulus	Ck0_cha	Pa
FOAM PLASTICITY		
Initial hydrostatic yield stress, compression	pc0	Pa

TABLE 2-29: HYPERELASTIC, ELASTOPLASTIC, PHASE TRANSFORMATION, AND DAMAGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Hardening function	pch	Pa
Hydrostatic yield stress, tension	pt	Pa
Initial uniaxial compressive yield stress	sigmauc0	Pa
Hardening function, uniaxial data	sigmauch	Pa
Plastic Poisson's ratio	bup	1
JOHNSON–COOK		
Strength coefficient	k_jcook	Pa
Hardening coefficient	n_jcook	1
Reference strain rate	epet0_jcook	1/s
Temperature exponent	m_jcook	
LUDWIK		
Strength coefficient	k_lud	Pa
Hardening exponent	n_lud	1
NONLINEAR ELASTIC FIBER		
Uniaxial stress function	saxa	N/m^2
PRESSURE-DEPENDENT PLASTICITY		
Yield function parameter	a1yield	1
Yield function parameter	a2yield	1/Pa
Yield function parameter	a3yield	Pa
Yield function parameter	a4yield	1/Pa
Potential parameter	a1potential	1
Potential parameter	a2potential	1/Pa
Potential parameter	a3potential	Pa
Potential parameter	a4potential	1/Pa
Yield function parameter	b1yield	Pa
Yield function parameter	b2yield	1
Yield function parameter	b3yield	Pa
Potential parameter	b1potential	Pa
Potential parameter	b2potential	1
Potential parameter	b3potential	Pa
Octahedral shape parameter	c1Lagioia	1
Octahedral shape parameter	c2Lagioia	1

TABLE 2-29: HYPERELASTIC, ELASTOPLASTIC, PHASE TRANSFORMATION, AND DAMAGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Octahedral shape parameter	c3Lagioia	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		
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
HILL'S COEFFICIENTS		
Hill's coefficients	{Hillcoefficients1, …, Hillcoefficients6}	$(m^2 \cdot s^4)/kg^2$
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		

TABLE 2-29: HYPERELASTIC, ELASTOPLASTIC, PHASE TRANSFORMATION, AND DAMAGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
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
BERGSTROM–BOYCE VISCOPLASTICITY		
Viscoplastic rate coefficient	A_BB	1/s
Flow resistance	sigRes_BB	N/m^2
Stress exponent	n_BB	1
Strain exponent	c_BB	1
Cutoff stress	sigmaco_BB	N/m^2
BERGSTROM–BISCHOFF VISCOPLASTICITY		
Viscoplastic rate coefficient	A_BeBi	1/s
Flow resistance	sigRes1_BeBi	N/m^2
Flow resistance	sigRes2_BeBi	N/m^2
Stress exponent	n1_BeBi	1
Stress exponent	n2_BeBi	1
Pressure hardening coefficient	a1_BeBi	1
Pressure hardening coefficient	a2_BeBi	1
BINGHAM VISCOPLASTICITY		
Viscosity	eta_bing	Pa·s
CHABOCHE VISCOPLASTICITY		

TABLE 2-29: HYPERELASTIC, ELASTOPLASTIC, PHASE TRANSFORMATION, AND DAMAGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Viscoplastic rate coefficient	A_cha	1/s
Reference stress	sigRef_cha	Pa
Stress exponent	n_cha	1
PARALLEL NETWORK VISCOPLASTICITY		
Viscoplastic rate coefficient	A_PN	1/s
Flow resistance	sigRes_PN	N/m^2
Stress exponent	n_PN	1
Stress exponent	c_PN	1
Cutoff stress	sigmaco_PN	N/m^2
Pressure hardening coefficient	a_PN	
PERIC VISCOPLASTICITY		
Viscoplastic rate coefficient	A_peric	1/s
Stress exponent	n_peric	1
PERZYNA		
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

TABLE 2-29: HYPERELASTIC, ELASTOPLASTIC, PHASE TRANSFORMATION, AND DAMAGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Reference shear strain	gammaRef	I
Strain exponent	n_strain	I
Bulk modulus in tension	Kt	Pa
Bulk modulus in compression	Kc	Pa
Ultimate deviatoric stress	q_ult	Pa
Ultimate strain	e_ult	I
ELASTOPLASTIC SOIL MATERIAL		
CAM-CLAY		
Swelling index	kappaSwelling	I
Compression index	lambdaComp	I
Void ratio at reference pressure	evoidref	I
Slope of critical state line	M	I
STRUCTURED CAM-CLAY		
Swelling index for structured clay	kappaSwellingS	I
Compression index for destructured clay	lambdaCompS	I
Void ratio at reference pressure for destructured clay	evoidrefS	I
Destructuring index for volumetric deformation	dvS	I
Destructuring index for shear deformation	dsS	I
Slope of critical state line	M	I
Additional void ratio at initial yielding	Deltaei	I
Initial structure strength	pbi	Pa
Plastic potential shape parameter	zetaS	I
Critical effective deviatoric plastic strain	epdevc	I
BARCELONA BASIC		
Swelling index	kappaSwelling	I
Swelling index for changes in suction	kappaSwellings	I
Compression index at saturation	lambdaComp0	I
Weight parameter	wB	I
Soil stiffness parameter	mB	Pa
Plastic potential smoothing parameter	bB	I
Tension to suction ratio	kB	I

TABLE 2-29: HYPERELASTIC, ELASTOPLASTIC, PHASE TRANSFORMATION, AND DAMAGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Void ratio at reference pressure and saturation	evoidref0	I
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	I
Bulk modulus in compression	Kc	Pa
Void ratio at reference pressure	evoidref	I
HYPERELASTIC AND VISCOELASTIC MATERIALS		
ARRUDA–BOYCE		
Macroscopic shear modulus	mu0	N/m^2
Number of segments	Nseg	I
BLATZ–KO		
Model parameters	phiBK	I
Model parameters	betaBK	I
Shear modulus	muBK	Pa
DELFINO		
Model parameters	aDelf	Pa
Model parameters	bDelf	I
EXTENDED TUBE		
Model parameters	GcET	Pa
Model parameters	GeET	Pa
Model parameters	alphaET	I
Model parameters	betaET	I
FUNG		
Coefficient matrix	AFung_iso (6-by-6 matrix)	I
Fung parameter c	cFung	Pa
GAO		
Model parameters	aG	Pa
Model parameters	nG	I
GENT		

TABLE 2-29: HYPERELASTIC, ELASTOPLASTIC, PHASE TRANSFORMATION, AND DAMAGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Macroscopic shear modulus	muG	Pa
Model parameter	jmG	I
HOLZAPFEL–GASSER–OGDEN		
Fiber dispersion	k3HGO	I
Fiber stiffness	k1HGO	Pa
Model parameter	k2HGO	I
MOONEY–RIVLIN		
Model parameters	C01, C02, C03, C10, C11, C12, C20, C21, C30	Pa
MURNAGHAN	The Murnaghan node adds three 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
POLYNOMIAL VOLUMETRIC BEHAVIOR		
Fourth-order bulk modulus	K2	Pa
Sixth-order bulk modulus	K3	Pa
VAN DER WAALS		
Chain network interaction	alphaW	I
Maximum chain stretch	lambdaW	I
Shear modulus	muW	Pa
Weight	betaW	I
VARGA		
Model parameters	c1VA	Pa
Model parameters	c2VA	Pa
YEOH		
Model parameters	c1YE	Pa
Model parameters	c2YE	Pa
Model parameters	c3YE	Pa
LAGOUDAS MODEL		

TABLE 2-29: HYPERELASTIC, ELASTOPLASTIC, PHASE TRANSFORMATION, AND DAMAGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Reference temperature	T0	K
Martensite start temperature	TMs	K
Martensite finish temperature	TMf	K
Austenite start temperature	TAs	K
Austenite finish temperature	TAf	K
Slope of martensite limit curve	CM	Pa/K
Slope of austenite limit curve	CA	Pa/K
Maximum transformation strain	etrmaxLagoudas	1
Calibration stress level	sigmaStar	N/m^2
LAGOUDAS MODEL, STRESS		
Martensite start stress	sMs	N/m^2
Martensite finish stress	sMf	N/m^2
Austenite start stress	sAs	N/m^2
Austenite finish stress	sAf	N/m^2
Measurement temperature	Tstress	K
LAGOUDAS MODEL, EXPONENTIAL LAW		
Initial maximum transformation strain	etrmin	1
Ultimate transformation strain	etrsat	1
Critical stress	scrit	N/m^2
Saturation exponent	kcrit	1/Pa
AUSTENITE PHASE		
Young's modulus	E_A	Pa
Heat capacity at constant pressure	Cp_A	J/(kg·K)
Thermal conductivity	k_A_iso	W/(m·K)
MARTENSITE PHASE		
Young's modulus	E_M	Pa
Heat capacity at constant pressure	Cp_M	J/(kg·K)
Thermal conductivity	k_M_iso	W/(m·K)
THERMAL EXPANSION, AUSTENITE PHASE		
Coefficient of thermal expansion	alpha_A_iso (3-by-3 matrix)	1/K

TABLE 2-29: HYPERELASTIC, ELASTOPLASTIC, PHASE TRANSFORMATION, AND DAMAGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Tangent coefficient of thermal expansion	alphatan_A_iso (3-by-3 matrix)	1/K
Thermal strain	dL_A_iso (3-by-3 matrix)	1
THERMAL EXPANSION, MARTENSITE PHASE		
Coefficient of thermal expansion	alpha_M_iso (3-by-3 matrix)	1/K
Tangent coefficient of thermal expansion	alphatan_M_iso (3-by-3 matrix)	1/K
Thermal strain	dL_M_iso (3-by-3 matrix)	1
SOUZA–AURICCHIO MODEL		
Reference temperature	TStar	K
Slope of limit curve	beta	Pa/K
Maximum transformation strain	etrmaxAuricchio	1
Elastic domain radius	sAf	N/m^2
Hardening modulus	Hk	N/m^2
Martensite start temperature	TMs_SA	K
Martensite finish temperature	TMf_SA	K
Austenite finish temperature	TAf_SA	K
Martensite start stress	sMs_SA	N/m^2
Martensite finish stress	sMf_SA	N/m^2
Austenite finish stress	sAf_SA	N/m^2
Measurement temperature	Tstress_SA	K
DAMAGE MATERIAL		
MAZARS DAMAGE FOR CONCRETE		
Tensile strength	sigmat	N/m^2
Fracture energy per area	Gft	J/m^2
Fracture energy per volume	gft	J/m^3
PHASE-FIELD DAMAGE		
Critical fracture stress	sigmacr	N/m^2
Critical energy release rate	Gc	J/m^2
Critical stress	scrit	N/m^2

TABLE 2-29: HYPERELASTIC, ELASTOPLASTIC, PHASE TRANSFORMATION, AND DAMAGE MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
SCALAR DAMAGE		
Peak strength	sigmap	N/m^2
Fracture energy per area	Gf	J/m^2
Fracture energy per volume	gf	J/m^3

Solid Mechanics Material Properties: Fatigue Module

These material property groups for material models in solid mechanics are used by the Fatigue Module.

TABLE 2-30: ELASTOPLASTIC AND FATIGUE BEHAVIOR MATERIAL PROPERTY GROUPS AND 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		

TABLE 2-30: ELASTOPLASTIC AND FATIGUE BEHAVIOR MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
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		
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 are used by the Geomechanics Module.

TABLE 2-31: GEOMECHANICS MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
BARCELONA BASIC		
Swelling index at saturation	kappaSwelling0	1
Swelling index for changes in suction	kappaSwellingsu	1
Compression index at saturation	lambdaComp0	1
Compression index for changes in suction	LambdaCompsu	1
Void ratio at reference pressure and saturation	evoidref0	1
Initial void ratio	evoid0b	1
Weight parameter	wb	1
Soil stiffness parameter	mb	Pa
Plastic potential smoothing parameter	bb	1
Tension to suction ratio	kb	1
Initial yield value for suction	sy0	Pa
CRITICAL STATE MODEL		
Poisson's ratio	nu	1
Shear modulus	G	Pa
Slope of critical state line	M	1
Slope of critical state line for plastic potential	Mq	1
Friction angle	internalphi	rad
Dilatation angle	psid	rad
Initial void ratio	evoid0	1
DRUCKER–PRAGER		
Drucker–Prager alpha coefficient	alphaDrucker	1
Drucker–Prager k coefficient	kDrucker	Pa
FRACTURE PARAMETERS		
Tensile fracture energy	Gft	J/m^2

TABLE 2-31: GEOMECHANICS MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Compressive fracture energy	Gfc	J/m^2
HARDENING SOIL		
Reference initial stiffness for primary loading	EiRef	Pa
Reference failure stiffness	E50Ref	Pa
Reference stiffness for unloading and reloading	EurRef	Pa
Small strain shear modulus	G0	Pa
Reference small strain shear modulus	G0Ref	Pa
Reference shear strain	gammaRef	1
Stress exponent	mh	1
Cohesion	c	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
MODIFIED CAM-CLAY		
Swelling index	kappaSwelling	1
Compression index	lambdaComp	1
Void ratio at reference pressure	evoidref	1
MODIFIED STRUCTURED CAM-CLAY		
Swelling index for structured clay	kappaSwellings	1
Compression index for destructured clay	lambdaCompd	1
Void ratio at reference pressure for destructured clay	evoidrefd	1

TABLE 2-31: GEOMECHANICS MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Initial void ratio for structured clay	evoid0s	I
Destructuring index for volumetric deformation	dv	I
Destructuring index for shear deformation	ds	I
Additional void ratio at preconsolidation pressure	Deltaec0	I
Initial structure strength	pb0	Pa
Plastic potential shape parameter	zeta	I
Plastic deviatoric strain at failure	edevepf	I
MOHR–COULOMB		
Cohesion	cohesion	Pa
Angle of internal friction	internalphi	rad
OTTOSEN		
Ottosen a parameter	aOttosen	I
Ottosen b parameter	bOttosen	I
Size factor	k1Ottosen	I
Shape factor	k2Ottosen	I
SOIL MATERIAL		
Initial strength parameter	ksoil0	Pa
Hardening function, strength	ksoilh	Pa
Friction parameter	asoil	I
Dilatancy parameter	aqsoil	I
Initial cohesion	cohesion0	Pa
Hardening function, cohesion	cohesionh	Pa
Friction angle	phis	rad
Dilatation angle	psis	rad
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 contains thermal expansion properties.

TABLE 2-32: THERMAL EXPANSION MATERIAL PROPERTY GROUPS AND 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
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.

Fluid Flow Material Properties: Inelastic Non-Newtonian Material Model

These material property groups for the inelastic non-Newtonian material models (including their associated physical properties) can be added to models from the **Material** window. All inelastic non-Newtonian models property groups are available with Polymer Flow Module. Power Law, Carreau, and Bingham–Papanastasiou property groups are also available with either CFD Module or Microfluidics Module.

TABLE 2-33: INELASTIC NON-NEWTONIAN MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
BINGHAM–PAPANASTASIOU		
Plastic viscosity	mu_p	Pa·s
Yield stress	tau_y	N/m^2
Local Property	m_p	s
CARREAU MODEL		
Zero shear viscosity	mu0	Pa·s

TABLE 2-33: INELASTIC NON-NEWTONIAN MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Infinite shear rate viscosity	mu_inf	Pa·s
Relaxation time	lam_car	s
Power index	n_car	1
CARREA U -YASUDA		
Zero shear viscosity	mu0cy	Pa·s
Infinite shear rate viscosity	mu_infcy	Pa·s
Relaxation time	lam_cy	s
Power index	n_cy	1
Transition parameter	a_car	1
CASSON–PAPANASTASIOU		
Plastic viscosity	mu_pc	Pa·s
Yield stress	tau_yc	N/m^2
Local Property	m_p	s
CROSS MODEL		
Zero shear viscosity	mu0c	Pa·s
Infinite shear rate viscosity	mu_infc	Pa·s
Power index	n_c	1
Critical stress	tau_tr	N/m^2
CROSS–WILLIAMSON		
Zero shear viscosity	mu0cw	Pa·s
Relaxation time	lam_cw	s
Power index	n_cw	1
DEKEE MODEL		
Shear rate viscosity parameter	mu_DK	Pa·s
Relaxation time	lam_DK	s
Yield stress	tau_DK	N/m^2
Local Property	m_p	s
ELLIS MODEL		
Zero shear viscosity	mu0e	Pa·s
Shear stress parameter	tau_half	N/m^2
Power parameter	a_e	1
HERSCHEL–BULKLEY–PAPANASTASIOU		
Fluid consistency coefficient	m_powhb	Pa·s

TABLE 2-33: INELASTIC NON-NEWTONIAN MATERIAL PROPERTY GROUPS AND PROPERTIES.

PROPERTY GROUP AND PROPERTY	NAME/VARIABLE	SI UNIT
Flow behavior index	n_powhb	1
Yield stress	tau_yhb	N/m^2
Local Property	m_p	s
Local Property	sr_ref	1/s
POWER LAW		
Fluid consistency coefficient	m_pow	Pa·s
Flow behavior index	n_pow	1
Local Property	sr_ref	1/s
Local Property	sr_lowlimit	1/s
ROBERTSON–STIFF–PAPANASTASIOU		
Fluid consistency coefficient	m_powrs	Pa·s
Flow behavior index	n_powrs	1
Yield stress	tau_yrs	N/m^2
Local Property	m_p	s
Local Property	sr_ref	1/s
SISKO		
Infinite shear rate viscosity	mu_infs	Pa·s
Fluid consistency coefficient	m_pows	Pa·s
Flow behavior index	n_pows	1
Local Property	sr_ref	1/s

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:

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

5 Under the **Definition** section:

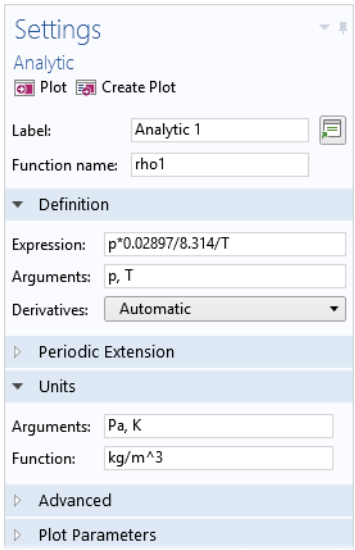
a In the **Expression** field, enter `p*0.02897/8.314/T`.

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

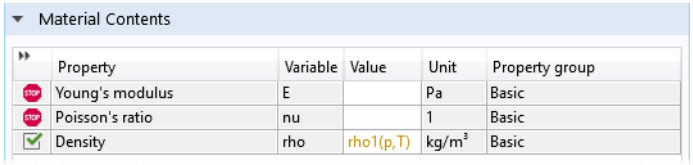
6 Under **Units**:

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 2-13.

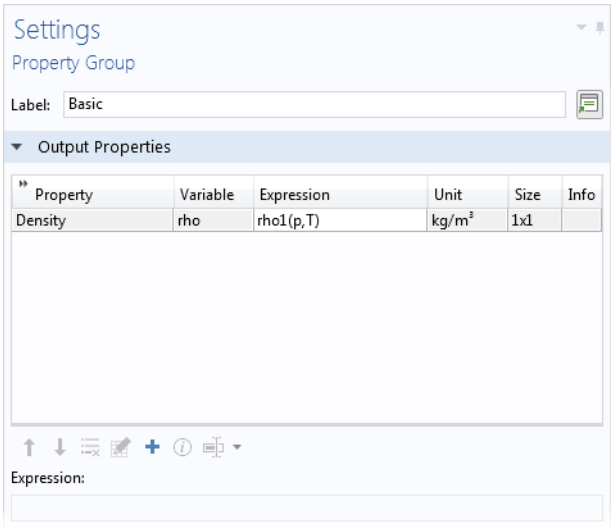


Figure 2-13: A density property is defined using an analytic function.

Index