

Composite Materials Module

User's Guide

Composite Materials Module User's Guide

© 1998–2018 COMSOL

Protected by patents listed on www.comsol.com/patents, and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 8,954,302; 9,098,106; 9,146,652; 9,323,503; 9,372,673; and 9,454,625. Patents pending.

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

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see www.comsol.com/trademarks.

Version: COMSOL 5.4

Contact Information

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

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

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

Part number: CM024701

C o n t e n t s

Chapter 1: The Composite Materials Module

About the Composite Materials Module	6
What Can the Composite Materials Module Do?	6
Where Do I Access the Documentation and Application Libraries?	7

Chapter 2: Composite Materials Modeling

About Composite Materials	10
Classification of Composites	10
Fiber-Reinforced Plastic (FRP) Composites	11
Benefits of Composites	12
Challenges of Composites	12
About Laminated Composite Shells	13
Fiber Arrangement	13
Stacking Sequence	13
Types of Laminate	15
Multi-Scale analysis	15
Micromechanics: Fiber-Matrix Modeling	17
Constituents of a Lamina	17
Homogenization and Cell Periodicity	18
Macromechanics: Laminate Modeling	19
Laminated Shell Theories	21
Equivalent Single Layer (ESL) Theory	21
Layerwise (LW) Theory	23
Selection of an Appropriate Laminate Theory	24

Layered Materials	25
Defining a Layered Material	25
Connecting a Layered Material to the Physical Geometry	28
Defining Laminare Coordinate System	29
Defining a Layered Material Stack	31
Saving and Loading Layered Material	32
Sweep over Layered Materials	33
 Layer Selections	 34
 Failure Modeling	 39
Failure Analysis	39
Buckling Analysis	40
Delamination Modeling	40
 Multiphysics Modeling	 41
Coupling with Electrical and Thermal Modeling	41
Coupling with Fluid and Acoustics Modeling	41
 Results Evaluation	 42
Layered Material Data Set	42
Layered Material Slice Plot	45
Through Thickness Plot	45

The Composite Materials Module

The Composite Materials Module is an optional add-on package for COMSOL Multiphysics® designed to assist you to model structural behavior that includes layered shells. The module is an add-on to the Structural Mechanics Module. The module is designed for researchers, engineers, developers, teachers, and students who want to simulate the behavior of layered structures, including multiphysics couplings.

With the Composite Materials Module you get the following extensions:

- The Layered Shell physics interface
- The Layered Linear Elastic Material in the Shell interface
- The **Thermal Expansion, Layered Shell** multiphysics coupling
- The **Layered Material** node under global materials, in which you specify the layup of a stacked material

About the Composite Materials Module

In this section:

- [What Can the Composite Materials Module Do?](#)
- [Where Do I Access the Documentation and Application Libraries?](#)

What Can the Composite Materials Module Do?

Using Composite Materials Module, a laminated composite shell, also known as composite laminate, which is an assembly of layers made of fibrous composite materials in order to provide required engineering properties e.g. in-plane stiffness, bending stiffness, shear stiffness, coefficient of thermal expansion etc. can be modeled. Different materials can also be used in different layers resulting in a hybrid laminate. In general, individual layers have orthotropic or transversely isotropic material properties making the full laminate as an anisotropic material.

It is possible to do multiscale analysis of a composite laminate involving micromechanical and macromechanical analysis. The micromechanical analysis deals with the modeling of individual lamina or its representative unit cell having fiber and matrix. The aim of this step is to compute the homogenized material properties of a lamina whereas the macromechanical analysis deals with the modeling of laminate having many layers. The aim here is to compute the response of a laminate under various loading conditions.

In COMSOL Multiphysics, composite laminates are analyzed either using Layerwise 3D Elasticity theory through the Layered Shell interface or using First Order Shear Deformation theory (ESL-FSDT) theory through the Layered Linear Elastic Material model in Shell interface.

In composite laminates, it is quite common to have different kinds of failure modes. Thus it becomes essential to perform various kinds of failure analysis for composite laminates. Both the laminate theories allow the computation of failure indices or safety factors based on the following criteria. The buckling of a composite laminate is a common phenomenon and also an important design criteria. So it is crucial to design a composite laminate which can withstand the compression or torsional buckling load. Also the composite laminate consists of many layers glued/laminated together. Under

different loading conditions especially impact loading, sometimes the bond between the two layers breaks in a specific region. The modeling of delaminated region becomes crucial in order to accurately predict the response of a damaged composite laminate.

Conventionally a composite laminate is designed based on the structural requirements however for an application where electrical and/or thermal effects are significant, it becomes crucial to design the laminate based on all structural, thermal, and electrical requirements. This poses a problem in a sense we need a tool which can actually solve for other physical phenomena especially electrical and thermal in addition to the structural mechanics. Using Composite Materials Module together with Heat Transfer and AC/DC Module, it is possible to simulate multiphysics models.

The composite laminates are defined on boundaries together with a layered material attached to it. Looking at the fact that composite laminates are anisotropic in nature and in general heterogeneous in through thickness direction, it becomes crucial to evaluate results in each layer as well as in through thickness direction apart from the results on the reference surface. There are various predefined tools which helps in the visualization of the results in a composite laminate.




In the *COMSOL Multiphysics Reference Manual*:

- [Studies and Solvers](#)
 - [The Physics Interfaces](#)
 - For a list of all the core physics interfaces included with a COMSOL Multiphysics license, see [Physics Interface Guide](#).
-

Where Do I Access the Documentation and Application Libraries?

A number of internet resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or

context-based) help, and the application libraries are all accessed through the COMSOL Desktop.



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

CONTACTING COMSOL BY EMAIL

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

COMSOL ACCESS AND TECHNICAL SUPPORT

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

COMSOL ONLINE RESOURCES

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

Composite Materials Modeling

This chapter introduces you to the modeling of composite materials.

In this chapter:

- [About Composite Materials](#)
- [About Laminated Composite Shells](#)
- [Micromechanics: Fiber-Matrix Modeling](#)
- [Macromechanics: Laminate Modeling](#)
- [Laminated Shell Theories](#)
- [Layered Materials](#)
- [Layer Selections](#)
- [Failure Modeling](#)
- [Multiphysics Modeling](#)
- [Results Evaluation](#)

About Composite Materials

A composite material is a heterogeneous material formed of two or more constituents integrated together to achieve enhanced structural performance. Owing to the improved strength and reduced weight when compared the conventional materials, the range of applications of composite materials spans across diverse fields. This necessitates a thorough understanding of the behavior of such materials.

Research has also begun to embed functionality like sensing, actuation, computation, and communication into composite materials. This is known as smart composite materials.

Some examples of engineered composite materials for various applications include:

- Fiber reinforced plastics
- Reinforced concrete beams
- Composite wood or plywood
- Ceramic matrix composites
- Metal matrix composites

Among different types of engineered composite materials, the use of fiber reinforced plastic (FRP) is growing rapidly. Some of the usage of fiber reinforced plastic material is in aircraft components, spacecraft components, wind turbine blades, automobiles, buildings, boat hulls, bicycles, and safety equipment.

This section includes:

- [Classification of Composites](#)
- [Fiber-Reinforced Plastic \(FRP\) Composites](#)
- [Benefits of Composites](#)
- [Challenges of Composites](#)

Classification of Composites

There are several ways to classify composite materials and one such way is to categorize them based on the type of its constituents, namely matrix and reinforcement.

Based on the type of matrix material, composite materials can be classified into the following categories:

- Polymer matrix composites (PMC)
- Metal matrix composites (MMC)
- Ceramic matrix composites (CMC)
- Carbon and graphite matrix composites (CGMC)

Based on the reinforcement shape, composite materials can be also classified into the following categories:

- Continuous fibers
- Random short fibers
- Flakes
- Whiskers
- Particulate

A commonly used fiber-reinforced plastic (FRP) material is a type of polymer matrix composites (PMC) in which continuous fibers are used as a reinforcement.

Fiber-Reinforced Plastic (FRP) Composites

A fiber-reinforced plastic (FRP) material typically constitutes of a fibrous part, which acts as the main load carrying element and surrounding matrix/resin to support the fiber and transfer the load. The fibers are arranged in a specified orientation in each layer (or lamina) of the material. A number of such laminae are stacked to form a laminated composite material which can be used for building a structural component.

The use of fiber composites in the manufacturing industry is increasing, as fiber composites are lighter and more corrosion resistant compared to traditional metallic engineering materials. Moreover, properties like strength, stiffness, and toughness can often be tailored to a specific application.

The fibers used for industrial applications are in general made of carbon, glass, aramid, or basalt. Based on the type of fiber material, the two popular FRPs available and typically used in the industry are:

- Carbon fiber reinforced plastics (CFRP)
- Glass fiber reinforced plastics (GFRP), also known as fiberglass

Benefits of Composites

Composite materials offer lot of benefits when compared to conventional metallic materials. Some of the key benefits are as follows:

- High strength to weight ratio
- High resistance to fatigue and corrosion degradation
- Improved friction and wear properties
- Low thermal conductivity and low coefficient of thermal expansion
- Possible to tailor material properties to meet the design requirements

Challenges of Composites

Looking at the fact that composite materials are man-made materials, there are also some challenges involved while using composite materials:

- High cost of raw materials and fabrication
- Difficulty in reuse and disposal
- Difficulty in joining different components
- Different types of failure modes
- Anisotropic materials — difficult to analyze

About Laminated Composite Shells

A laminated composite shell, also known as composite laminate, is an assembly of layers made of fibrous composite materials in order to provide required engineering properties e.g. in-plane stiffness, bending stiffness, shear stiffness, coefficient of thermal expansion etc. Different materials can also be used in different layers, resulting in a hybrid laminate. In general, individual layers have orthotropic or transversely isotropic material properties making the full laminate an anisotropic material.

This section includes:

- [Fiber Arrangement](#)
- [Stacking Sequence](#)
- [Types of Laminate](#)
- [Multi-Scale analysis](#)

Fiber Arrangement

Depending on the way fibers are reinforced/arranged in a matrix material, the composite layers/plies can be categorized as:

- Unidirectional ply
- Woven fabric

Woven fabrics are produced by the interlacing of warp (0°) fibers and weft (90°) fibers in different weave style. Woven fabric can be in biaxial, triaxial, or multiaxial configurations, and can also have different types of weaving, namely planar weaving or 3D weaving.

Stacking Sequence

A composite laminate is defined as stacking of two or more unidirectional layers/plies/laminae with same or different fiber orientation with respect to the global direction. The laminae may be made of same or different materials and can have individual thicknesses.

The stacking sequence, as shown in [Figure 3-1](#), essentially is the fiber orientation in each ply with respect to the first axis of the laminate coordinate system. The stacking

sequence is interpreted from bottom to top and orientation angles are in general specified in degrees.

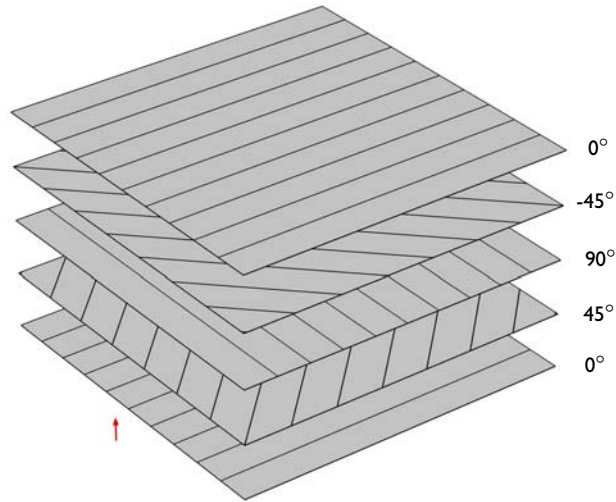


Figure 3-1: Example of a stacking sequence $[0/45/90/-45/0]$ used in a composite laminate.

The designation of stacking sequence is specified in a concise form especially to accommodate laminates having large number of layers leveraging the fact that most of the laminates have either symmetry or many layers with same orientation.

Some examples of stacking sequence designations are given in the table below:

TABLE 3-1:

STACKING SEQUENCE	DESCRIPTION	LAYERS
$[45/-30/0]$	One layer each of 45° , -30° , and 0°	3
$[45/-30/0]_s$	Symmetric with 45° , -30° , and 0° layers	6
$[45/-30/0]_3$	Three sets of 45° , -30° , and 0° layers	9
$[(45)_2/-30/(0)_3]_s$	Symmetric with 45° , 45° , -30° , 0° , 0° , and 0° layers	12
$[45/-30/0]_{as}$	One layer each of 45° , -30° , 0° , 0° , 30° , -45° layers (as = antisymmetric)	6

Types of Laminate

Based on the stacking sequence, composite laminates can be classified into following categories:

Angle-ply Laminate

A laminate is called *angle-ply laminate* if it has layers of the same thickness and material and are oriented at θ and $-\theta$.

Cross-ply Laminate

A laminate is called *cross-ply laminate* if all the layers used to fabricate the laminate have the orientation 0° and 90° .

Balanced Laminate

A laminate is called *balanced laminate* when it has pairs of layers with same thickness and material and the angles of layers have opposite signs. Balanced laminates can also have layers at 0° and 90° .

Symmetric Laminate

A laminate is called *symmetric* when the material, angle, and thickness of the layers are the same above and below the midplane.

Antisymmetric Laminate

A laminate is called *antisymmetric* when the material and thickness of the layers are same above and below the midplane, but the orientation of the layers have opposite signs above and below the midplane.

Multi-Scale analysis

The analysis of a composite laminate is comparatively difficult as it involves different geometric scales. The analysis performed at two different scales are as follows:

MICROMECHANICS ANALYSIS

This step deals with the modeling of individual lamina or its representative unit cell having fiber and matrix. The aim of this step is to compute the homogenized material properties of a lamina.

MACROMECHANICS ANALYSIS

This step deals with the modeling of laminate having many layers. The homogenized material properties of a lamina computed from micromechanics analysis is the input to

this analysis. The aim here is to compute the response of a laminate under various loading conditions.

Micromechanics: Fiber-Matrix Modeling

While analyzing components made up of laminated composites, it becomes essential to estimate the homogenized material properties of the lamina first. The task of computing the homogenized material properties of a lamina using the representative unit cell having fiber and matrix materials is known as micromechanics analysis.

Constituents of a Lamina

A lamina, in general, consists of fiber and matrix materials. The homogenized properties of a lamina depends on the constituents properties as well as the volume fraction in which constituents are mixed. The fiber material is mostly an orthotropic material whereas the matrix material is an isotropic material.

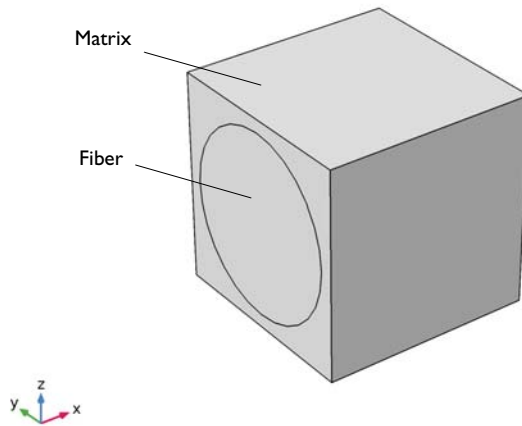


Figure 3-2: Example of a representative volume element (RVE) or unit cell of a lamina having fiber and matrix materials.

A representative volume element (RVE) or unit cell is the smallest volume element of a lamina which can be repeated to create a full lamina. An example of a representative volume element is shown in the [Figure 3-2](#). In this case, the length of the RVE in the

fiber direction is immaterial, while the size in the two transverse directions is given by the fiber spacing.

Homogenization and Cell Periodicity

In order to perform a micromechanics analysis of a composite lamina and compute its homogenized orthotropic material properties, the **Cell Periodicity** node in the Solid Mechanics interface is used.

In order to extract the homogenized material properties, the **Cell Periodicity** node performs the following sequence of operations:

- Applies periodic boundary condition to three pairs of faces of the unit cell.
- Creates a stationary study with six different unit load cases.
- Solves load cases and obtains the components of homogenized elasticity matrix of a lamina from the computed stress-strain relations.
- Creates a new homogeneous material node under **Global Definitions->Materials** and assigns the computed homogenized elasticity matrix to it.

This new homogeneous material can now be used as an input to the macromechanics analysis.



In the *Structural Mechanics Module User's Guide*:

- [Effective Properties of Periodic Structures](#)
 - [Cell Periodicity](#)
 - [Periodic Cell Theory](#)
-

Macromechanics: Laminate Modeling

The task of modeling a composite laminate and computing its response for given loading conditions is known as macromechanics analysis. In this step, the homogenized material properties of a lamina computed using micromechanics analysis are directly used.

In order to model a composite laminate, the following properties need to be specified:

- Number of layers
- Homogenized material properties of each layer
- Orientation of the principal material directions
- Thickness of each layer
- Stacking sequence

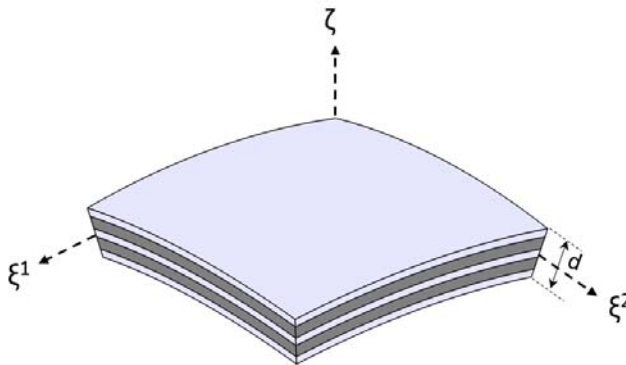


Figure 3-3: 3D representation of a composite laminate showing the total laminate thickness and the laminate coordinate system.

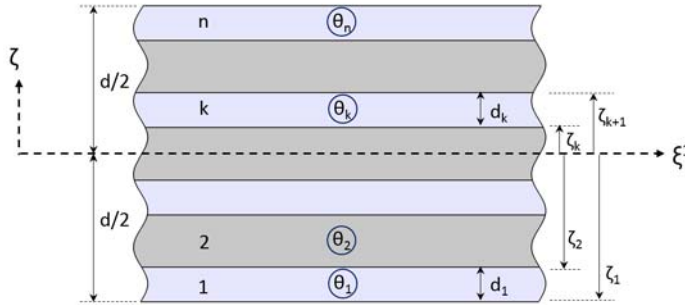


Figure 3-4: Through thickness (2D) representation of a composite laminate with n layers indicating thickness and fiber orientation of each layer. The position of the reference surface with respect to the laminate is also shown.

In order to attach the layer definition to geometric surfaces, the following properties need to be defined:

- **Laminate coordinate system:** These are the global directions for a composite laminate about which the stacking sequence is interpreted as shown in [Figure 3-3](#).
- **Reference surface position:** This is the position of the reference surface or geometric surface with respect to the midplane of the laminate as shown in [Figure 3-4](#).

Once all above definitions are specified, a regular finite element modeling approach can be used. It involves setting up the physics (loads, boundary conditions), creating a finite element mesh, performing various types of analysis, including static, dynamic, failure, buckling, and multiphysics analysis.

Laminated Shell Theories

Analysis of laminated composite shells is commonly based on one of three different theories:

- 1 Equivalent single layer (ESL) theory
 - a Classical laminated plate theory (CLPT)
 - b First order shear deformation laminated plate theory (FSDT)
 - c Higher order shear deformation laminated plate theory
- 2 Three dimensional elasticity theory
 - a Traditional 3D elasticity theory
 - b Layerwise 3D elasticity theory
- 3 Multiple model methods

In COMSOL Multiphysics, composite laminates are analyzed either using layerwise 3D elasticity theory through the Layered Shell interface or using first order shear deformation theory (ESL-FSDT) theory through the **Layered Linear Elastic Material** in Shell the interface.

This section includes:

- [Equivalent Single Layer \(ESL\) Theory](#)
- [Layerwise \(LW\) Theory](#)
- [Selection of an Appropriate Laminate Theory](#)

Equivalent Single Layer (ESL) Theory

Composite laminates are formed by stacking layers of different materials and/or having different fiber orientations. In general their planar dimensions are two orders of magnitude larger than their thickness. Often laminated structures are used in applications requiring high membrane and bending strengths. Therefore, in many cases composite laminates can be modeled using a shell element based on an equivalent single layer theory.

This is a classical way of modeling composite laminates. This theory treats a heterogeneous laminated composite as a statically equivalent single layer. This theory reduces a 3D continuum problem to an equivalent 2D problem, thus reducing the size and computational time of the problem.

In addition to simplicity and low computational cost, this theory provides sufficiently accurate description of the global response for a thin to moderately thick laminates such as gross deflections, critical buckling loads, and eigenfrequencies with corresponding mode shapes.

CLASSIFICATION

The ESL theories can be classified into various groups based on the description of the transverse shear stresses.

Classical Laminate Plate Theory (CLPT)

The classical laminate plate theory is an extension of Kirchhoff or classical plate theory used for single layer thin shells. In this theory, transverse shear stresses are neglected and the deformation is entirely due to the bending and in-plane stretching.

First Order Shear Deformation Theory (FSDT)

The first order shear deformation theory is similar to the Mindlin-Reissner shell theory used for single layer thick shells. This theory extends the kinematics of CLPT by including the gross transverse shear deformation. The transverse shear strain is assumed to be constant with respect to the thickness coordinate. As the transverse shear strain has a constant value, this theory requires a shear correction factor.

Higher Order Shear Deformation Theory

This theory is the extension of FSDT and here the displacement field is approximated in such a way that the transverse shear strain varies quadratically with respect to the thickness coordinate. It makes the transverse shear stresses zero at the top and bottom surfaces of the laminate and thus eliminates the need of a shear correction factor.

FIRST ORDER SHEAR DEFORMATION THEORY (FSDT)

In COMSOL Multiphysics, first order shear deformation (ESL-FSDT) theory is one of the options for analyzing composite laminates. This theory is implemented in the **Layered Linear Elastic Material** model in Shell interface. It has an MITC (*mixed interpolation of tensorial components*) formulation. As this theory accounts for the transverse shear deformation, it can be used for rather thick shells.

KEY FEATURES

- Degrees of freedom (3 displacements, 3 rotations) are defined only at the midplane or reference plane of the composite laminate.
- Suitable for modeling thin to moderately thick laminates.
- Suitable for finding global response of the laminate e.g. gross deflections, eigenfrequencies, critical buckling load etc.

- Not very computationally expensive, and thus suitable for analysis of laminates having large number of layers.
- Requires shear correction factor for thicker laminates where transverse shear stresses are not negligible



In the *Structural Mechanics Module User's Guide*: [Theory for FSDT Laminated Shell](#)

Layerwise (LW) Theory

A layerwise (LW) theory for modeling composite laminates is very similar to a traditional 3D elasticity theory, where the degrees of freedom are only the displacement fields defined in the product geometry created by the reference surface and an extra dimension in the thickness direction.

There are two approaches depending on the way degrees of freedom are defined:

- Partial displacement field approach
- Full displacement field approach

In the partial displacement field approach, the laminate thickness remains constant, whereas the full displacement field approach allows a change in thickness of the laminate. In COMSOL Multiphysics, a full displacement field approach based layerwise theory is implemented in Layered Shell interface.

The layerwise theory is more accurate than the equivalent single layer theory, but it comes with the cost of having more degrees of freedom. It is significantly more expensive in terms of computer resources.

From accuracy point of view, layerwise theory is as accurate as traditional 3D elasticity theory, but it has several benefits over traditional 3D elasticity theory:

- No need to build a 3D geometry with many thin layers.
- Easy to handle layerwise and interfacial data.
- In-plane finite element meshing is independent of the out-of-plane (thickness direction) meshing.
- A separate, either lower or higher, shape function order can be chosen in the thickness direction in order to avoid shear locking or to gain accuracy advantages.

KEY FEATURES

- Degrees of freedom (3 displacements) are defined in the product geometry created by the reference surface and an extra dimension in the thickness direction.
- Predicts correct through-thickness or transverse shear stress variation, thus making it suitable for modeling thick shells.
- Predicts correct inter-laminar stresses and has degrees of freedom in the thickness direction making it suitable for delamination and detailed damage analysis.
- Supports non-linear material models for advanced composite modeling.
- Supports different material models in different layers of a laminate. It is for example possible to combine linear elastic and hyperelastic materials in a single laminate.

Selection of an Appropriate Laminate Theory

In the above two sections, the key features of both equivalent single layer and layerwise theory have been outlined. Based on that, an appropriate laminate theory can be chosen for a particular problem. Without going into more details, the laminate aspect ratio (the ratio between in-plane size and thickness) gives an important hint about the theory that would be more suitable for a particular laminate.

In [Figure 3-5](#), it can be seen that ESL theory is more suitable for moderately thick to very thin laminates, whereas layerwise theory is more suitable for moderately thin to very thick laminates. The selection of ESL theory for very thick laminates may give rise to incorrect transverse shear stresses whereas the selection of layerwise theory for very thin laminates may give rise to shear locking.

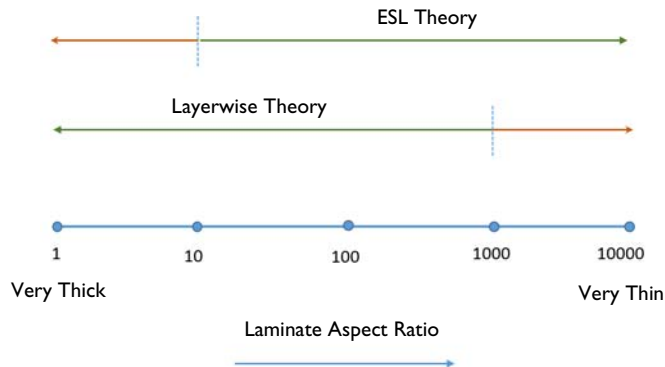


Figure 3-5: Selection of an appropriate laminate theory based on the laminate aspect ratio.

Layered Materials

The technique for defining material data and other properties for a layered shell is somewhat different from what is common in most other physics interfaces. In general, several different materials are used on the same boundary, so it would not be possible to use the assignment in a normal **Material** node in the component.

Rather, you defined the whole laminate, including materials, layer thickness, orientations etc. as one unit. The workflow is described in this section.

This section includes:

- [Defining a Layered Material](#)
- [Connecting a Layered Material to the Physical Geometry](#)
- [Defining Laminate Coordinate System](#)
- [Defining a Layered Material Stack](#)
- [Saving and Loading Layered Material](#)
- [Sweep over Layered Materials](#)

Defining a Layered Material

Start by adding a **Layered Material** at the **Materials** node under **Global Definitions**. Then, take the following steps to define the laminate:

- Add the required number of layers in the **Layer Definition** section of a **Layered Material** node.
- Define the referenced materials under the **Materials** node. You can either add a **Blank Material** and fill in the required material properties or choose a material from the **Material Library**.
- Select the appropriate material in each layer.
- Define the rotation angle (in degrees) for each layer. This is the orientation of the fiber direction with respect to the principal laminate coordinate system.
- Define the thickness of each layer.
- By default, two mesh elements in each layer are used which means there are two sublayers in each material layer. Optionally, you may want to either increase or

decrease the number of mesh elements in an individual layer in order to improve the accuracy or reduce the computation time.



Note that required material properties for the material in a certain layer depends on the **Solid Model** chosen in the **Linear Elastic Material** node in Layered Shell or **Layered Linear Elastic Material** node in Shell interface.

- For the **Isotropic** model (with Young's modulus and Poisson's ratio option), scalar values of Young's modulus and Poisson's ratio are required.
- For the **Orthotropic** model, Young's modulus, shear modulus, and Poisson's ratio along the principal material directions is required.
- For the **Anisotropic** model, the full homogenized elasticity matrix of along the principal material directions is required.



INTERFACE MATERIALS

By default, no material is selected on the interface between two layers in the through-thickness direction. All the layers are assumed in a continuity state.

Optionally, in order to model a very thin layer of material, for example a glue layer between the two material layers, a material can be added in **Interface Properties** section. If an interface material is defined, you can create a discontinuity between two layers and use the interface material properties. To do this, add a **Thin Elastic Layer, Interface** node in the Layered Shell interface. All interfaces selected in that node should have a material assignment, but you there is no need to assign a material to interfaces which are not affected by such special features.

PREVIEW PLOTS

As the physical geometry is only the boundaries, it becomes important to visualize the layup, including stacking sequence and thickness values for each layer.

You can do that by creating **Layer Cross Section Preview** () and **Layer Stack Preview** () plots. An example of a layer cross section plot showing a 2D cross section of a laminate having is shown in [Figure 3-6](#). The thickness of each layer is visible. An example of a layer stack preview plot showing the stacking sequence is shown in [Figure 3-7](#).

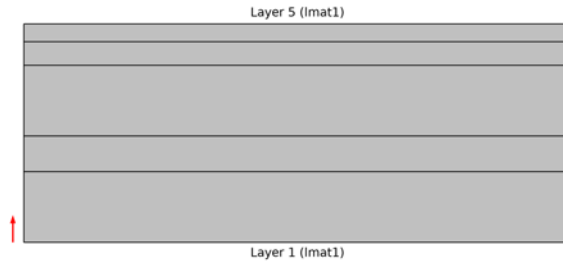


Figure 3-6: Example of a layer cross section preview plot showing a composite laminate having 5 layers with different thickness values.

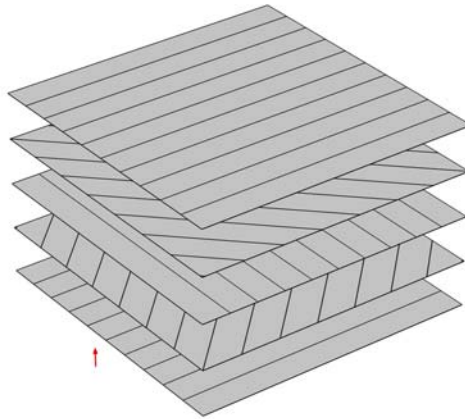


Figure 3-7: Example of a layer stack preview plot showing a composite laminate having 5 layers with stacking sequence $[0/45/90/-45/0]$.

Note that in the layer stack preview plot, the distance between any two layers are always the same and it does not reflect the actual thickness of the layers.

Since the preview plots are used for variety of composite laminates having different combination of number of layers, thickness values, and stacking sequence, it is possible to customize the plot settings in terms of changing the thickness to width ratio or the distance between the orientation lines. It is also possible to switch off the orientation lines and labels.



In the *COMSOL Multiphysics Reference Manual*:

- [Layered Material](#)
-

Connecting a Layered Material to the Physical Geometry

After creating the layered material definition, the next step is to attach the layer definition to the physical geometrical surfaces. The most fundamental way of doing that is through a **Layered Material Link** node, which you add under the **Materials** node within a component.

In that node, you select the boundaries where you want to attach the layer definition. The position of the geometric boundaries with respect to laminate definition is also defined. You can attach the boundary to the laminate midplane, down side, up side, or at a user defined location as shown in [Figure 3-8](#).

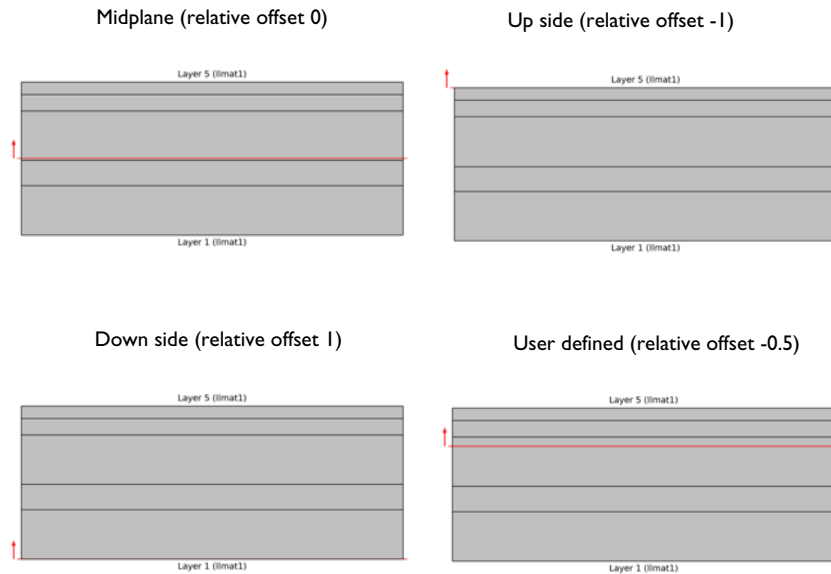


Figure 3-8: Position of geometric boundaries with respect to the composite laminate.

Defining Laminate Coordinate System

The next step is to define a coordinate system for composite laminate in which the stacking sequence is interpreted. You can either customize the default **Boundary System** or add a new instances of boundary systems under the **Definitions** node within a component. Select the coordinate system in the **Layered Material Link** node.

In order to create a suitable laminate coordinate system using **Boundary System** node,

- Define an appropriate direction from which first tangent direction of the coordinate system is obtained
- Reverse the normal direction if needed

Some examples of laminate coordinate systems in a cylindrical composite laminate created using different settings in **Boundary System** are given below.

Example 1

The first axis of the laminate coordinate system is aligned with the global x direction. This can be created by setting the **Create first tangent direction from** option to **Global Cartesian (spatial)** and **Axis** to x .

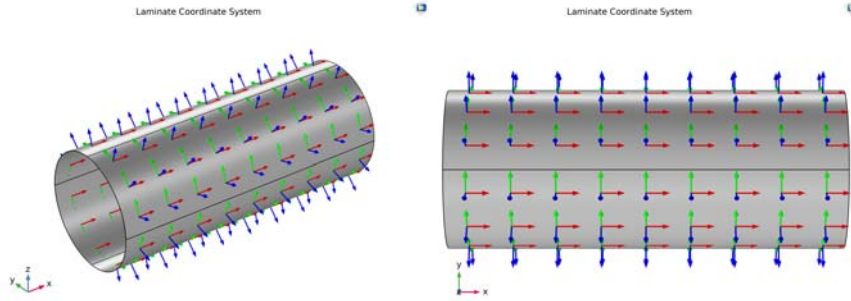


Figure 3-9: Example of a laminate coordinate system when the first axis is aligned with global x direction.

Example 2

The first axis of the laminate coordinate system is at 45° to the global x direction. This can be created by first defining a **Cylindrical System** having **Longitudinal axis** as x -axis and then setting the **Create first tangent direction from** to **Cylindrical System** and **Axis** to **Manual** with $\{0,-1,1\}$ as orientation.

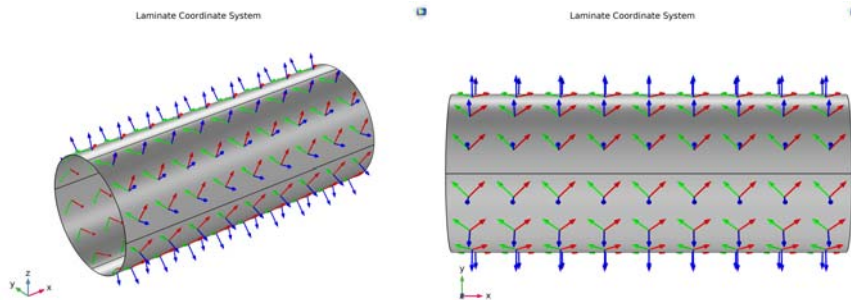


Figure 3-10: Example of a laminate coordinate system when the first axis is at 45° to the global x direction.

Example 3

The first axis of laminate coordinate system is aligned with the global x direction but with the normal direction is pointing inward. This coordinate system can be created in a same way as in the first example. In addition to that, the **Reverse Normal** direction

option is selected in the **Boundary System** node in order to point the normal vector inward.

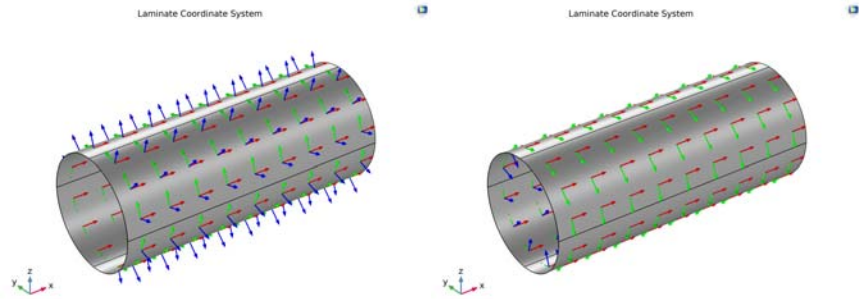


Figure 3-11: Example of a laminate coordinate system where the first axis is aligned with the global x direction and the normal vector is pointing outward and inward respectively.



Note that it is also possible to optionally reverse the normal direction only on the few selected boundaries by using the **Reverse Normal** subnode on the **Boundary System** node.



In the *COMSOL Multiphysics Reference Manual*:

- [Layered Material Link](#)
- [Boundary System](#)

Defining a Layered Material Stack

There is often a situation when different layered materials are stacked on top of each other to build a composite laminate for example a laminate having the stacking sequence $[A/B/C/B/A]$ where A, B, C are three different layered materials or sub-laminates.

To simplify such modeling, use the **Layered Material Stack** node, which can be added under the **Materials** node within a component. In **Layered Material Stack**, you define the

position and laminate coordinate system similar to in a **Layered Material Link**. You do however get a possibility to link multiple layered materials in the stacking order.



Note that it is also possible to add a **Layered Material** node directly under the **Layered Material Stack** node.

An example of a layered material stack having three different layered materials is shown in [Figure 3-12](#).



Figure 3-12: Example of a layered material stack having three different (A,B,C) layered materials.



In the *COMSOL Multiphysics Reference Manual*: [Layered Material Stack](#)

Saving and Loading Layered Material

After setting up a layered material definition together with the material properties of each layer, you may want to save it in order to reuse the data in a different model. This can be achieved in two different ways:

- Add the **Layered Material** to a **User-Defined Library** by right-clicking and choosing this option. This adds the layer definition as well as the material properties of each layer to the user-defined library.
- Save the model file with only the layered material set-up.

Next time when you want to use the same layered material setup, then either you can add it from a **User-Defined Library** if it is available there, or you can browse to the model file in which layered material is stored by following these steps:

- Right click on **Materials** node under **Global Definitions** and choose **Browse Materials**.
- Choose **Import Material Library** and browse to the model file containing the layered material definition.
- Choose a layered material from the newly added material library and add it to the **Global Materials** in the model. Alternatively, you can add it to the **User-Defined Library**.

Sweep over Layered Materials

In many cases, a primary goal while designing a composite laminate is to optimize the layered material definition which includes number of layers, arrangement of layers, material properties of each layer, thickness of each layer, and stacking sequence.

In order to do this, you can add multiple **Layered Materials** nodes under a **Switch** node. You add the **Switch** node from **Materials** under **Global Definitions**. The **Switch** node can be selected in the **Layered Material Link** node.

With these settings, it is possible to add a **Material Sweep** node to any study in order to solve for all layered materials.

Layer Selections

Once a layered material is defined, it is possible to use the layer information while setting up the selections in a physics interface and its different features.

The physics interface and its features have a section named **Layer Selection**. The layer selections interact with the standard selections of geometrical objects (boundary, edges, or points) in order to provide a complete specification of where a material property or boundary condition is to be applied.

The physics features can be of two, fundamentally different, types: Those who act on the layers themselves, and those who act on the interfaces between layers. The settings for these two types of features are slightly different.

The default selections in the **Layer Selection** section differ between different physics features. This reflects the fact that some physics features (such as thermal expansion) are more likely to be applied to all layers, whereas other features (such as added mass) are more likely to be used for a single layer or interface.

In the **Layer** drop-down list, you can select

- **All layered materials**. This option is not available for all features.
- Any **Layered Material Link** defined under **Materials** in the current component.
- Any **Layered Material Stack** defined under **Materials** in the current component.
- Any **Single Layer Material** defined under **Materials** in the current component.

If you select **All layered materials**, you cannot control individual layers; the contribution is given to all layers. All information is taken from the definitions made in the layered material features (**Layered Material Link**, **Layered Material Stack**, or **Single Layer Material**) under **Materials** in the current component. This means that a single physics node can accommodate several different stacking sequences. As geometrical selection, you can use any object selected in any of the layered material features.

If you select an individual **Layered Material Link**, **Layered Material Stack**, or **Single Layer Material**, then you can only select geometrical objects which are part of the selections of that feature. In most cases, you will get access to a list of check boxes where you can further limit the contributions to individually selected layers or interfaces to which this contribution is to be added.

LAYER SELECTIONS

When a physics feature acts on layers, the interpretation of layer selections are:

- A boundary selection in the geometry corresponds to a domain contribution in the selected layers.
- An edge selection in the geometry corresponds to boundary contributions on the faces of the layers exposed to that edge.
- A point selection in the geometry corresponds to edge contributions on corners of the selected layers meeting at the selected point.

This is illustrated in [Figure 3-13](#) to [Figure 3-15](#) where the patch at the lower left is the actual shell, and the 3D sketch shows an expanded view of what it would represent in the physical world.

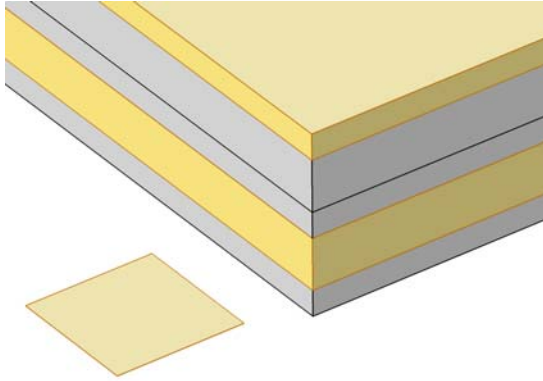


Figure 3-13: Boundary selection with layers 2 and 5 selected.

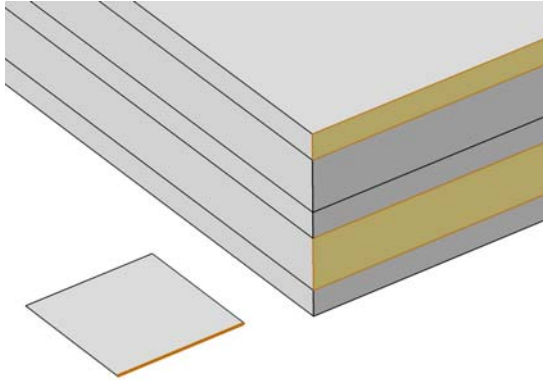


Figure 3-14: Edge selection with layers 2 and 5 selected.

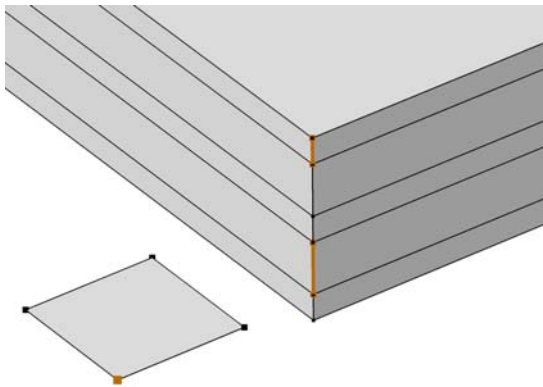


Figure 3-15: Point selection with layers 2 and 5 selected.

INTERFACE SELECTIONS

When a physics feature acts on interfaces, the interpretation of interface selections are:

- A boundary selection in the geometry corresponds to boundary contributions on the selected interfaces.

- An edge selection in the geometry corresponds to edge contributions on the interface edges parallel to the selected edge.
- A point selection in the geometry corresponds to point contributions on the interface corners located at the selected point.

This is illustrated in [Figure 3-16](#) to [Figure 3-18](#) where the patch at the lower left is the actual shell, and the 3D sketch shows an expanded view of what it would represent in the physical world.

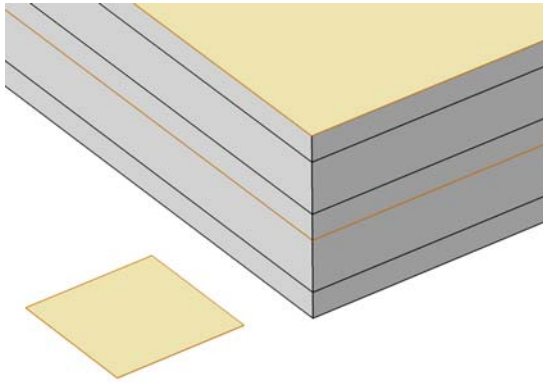


Figure 3-16: Boundary selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.

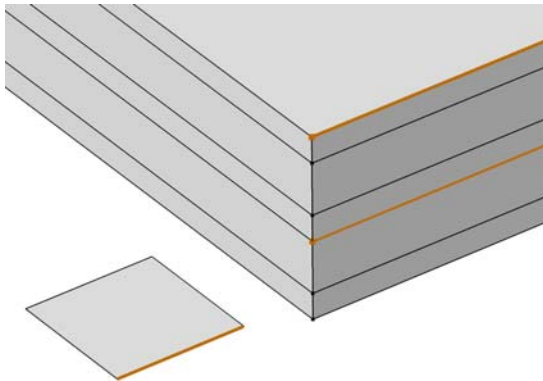


Figure 3-17: Edge selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.

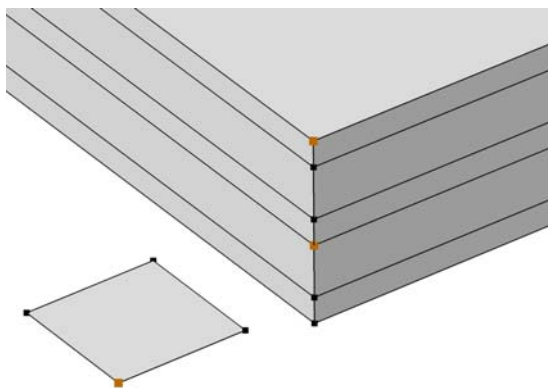


Figure 3-18: Point selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.

Failure Modeling

In composite laminates, it is quite common to have different kinds of failure modes. Thus it becomes essential to perform various kinds of failure analysis for composite laminates.

In this section:

- [Failure Analysis](#)
- [Buckling Analysis](#)
- [Delamination Modeling](#)

Failure Analysis

In COMSOL Multiphysics, both laminate theories allow computation of failure indices or safety factors based on the following criteria:

- von Mises Isotropic
- Tresca Isotropic
- Rankine Isotropic
- St. Venant Isotropic
- Jenkins Orthotropic
- Waddoups Orthotropic
- Tsai-Hill Orthotropic
- Hoffman Orthotropic
- Tsai-Wu Orthotropic
- Tsai-Wu Anisotropic

In addition to that, the ESL theory also allows the following failure criteria:

- Azzi-Tasi-Hill Orthotropic
- Norris Orthotropic
- Modified Tsai-Hill Orthotropic

To compute safety factors, you add a **Safety** node under the material model.



In the *Structural Mechanics Module User's Guide*:

- The [Safety](#) node
- [Safety Factor Evaluation](#)

Buckling Analysis

The buckling of a composite laminate is a common phenomenon and also an important design criterion. It is crucial to design a composite laminate which can withstand the compression or torsional buckling load.

In COMSOL Multiphysics, both the laminate theories can be used for computing the critical buckling load factor of a composite laminate.



In the *Structural Mechanics Module User's Guide*: [Linearized Buckling Analysis](#)

Delamination Modeling

A composite laminate consists of many layers glued or laminated together. Under different loading conditions, particularly impact loading, the bond between the two layers can break in a specific region. Modeling of the delaminated region becomes crucial in order to accurately predict the response of a damaged composite laminate.

In COMSOL Multiphysics, this can be achieved using the layerwise theory based Layered Shell interface. There, the **Thin Elastic Layer, Interface** node can be used for modeling the delaminated region between different layers of a composite laminate.



In the *Structural Mechanics Module User's Guide*: [Thin Elastic Layer, Interface](#)

Multiphysics Modeling

In most cases, a composite laminate is designed based on the structural requirements, but there are some cases where also other physical phenomena must be taken into consideration. There are two fundamentally different types of interaction between the mechanics in the laminate and other processes:

- Some physical processes occur inside the laminate. Examples of this is heat flux, electric currents, or diffusion processes.
- In other cases, the laminate acts as a boundary for some 3D domain where something important occurs. A prototype for this is fluid flow.

Coupling with Electrical and Thermal Modeling

In some applications where electrical or thermal effects are significant, it can be important to design the laminate based on the combined response. The layered material technology in COMSOL Multiphysics allows you to use different physics interfaces in the same layered material, and it is possible to solve for all the physical phenomena simultaneously, including couplings between them.

From structural design point of view, both laminate theories can be coupled to electrical and thermal modeling of composite laminates. One such example is Joule heating and thermal expansion of a composite laminate.

Coupling with Fluid and Acoustics Modeling

Models based on the ESL theory allow you to couple the composite laminate with a surrounding fluid for fluid-structure interaction or acoustic applications.

Results Evaluation

Composite laminates are defined on boundaries together with a layered material attached to it. As composite laminates are anisotropic in nature and in general heterogeneous in the through-thickness direction, evaluating results on the reference surface only is usually not sufficient. You will need to evaluate results in each layer, as well as in the through-thickness direction.

In addition to the standard result presentation tools, COMSOL Multiphysics offers specialized methods for evaluating and plotting results in composite laminates:

- [Layered Material Data Set](#)
- [Layered Material Slice Plot](#)
- [Through Thickness Plot](#)

Layered Material Data Set

Even though the composite laminate is modeled as a surface (2D) geometry, you can visualize the results on a geometry with finite thickness. For this, you use the **Layered Material** data set.

Often the aspect ratio of a laminate is high, so it is difficult to visualize the variations in the thickness direction. For this reason, the **Layered Material** data set provides an option to scale the geometry in the thickness direction.

Using this data set, you can plot the results in two ways:

- Plots can be done on a full 3D geometry.
- You can create slices in the shell at different levels. The slices can be placed at either mesh nodes in the thickness direction, at layer midplanes, or at interfaces.

Below is an example of how a thin composite laminate with 2 layers can be visualized. The geometry is a cylinder which is viewed along its axis. In [Figure 3-19](#), the following evaluation and plotting modes are shown:

- Case A: The geometry. This is what you see if you do not use the specialized data set.
- Case B: Layered material data set with true thickness (the default). It gives a solid representation, but you would need to zoom in to see details.
- Case C: Layered material data set with 20x scaling in thickness direction.

- Case D: Layered material data set with 20x scaling in thickness direction and slices at mesh nodes.
- Case E: Layered material data set with 20x scaling in thickness direction and slices at layer midplanes.
- Case F: Layered material data set with 20x scaling in thickness direction and separate slices at interfaces.

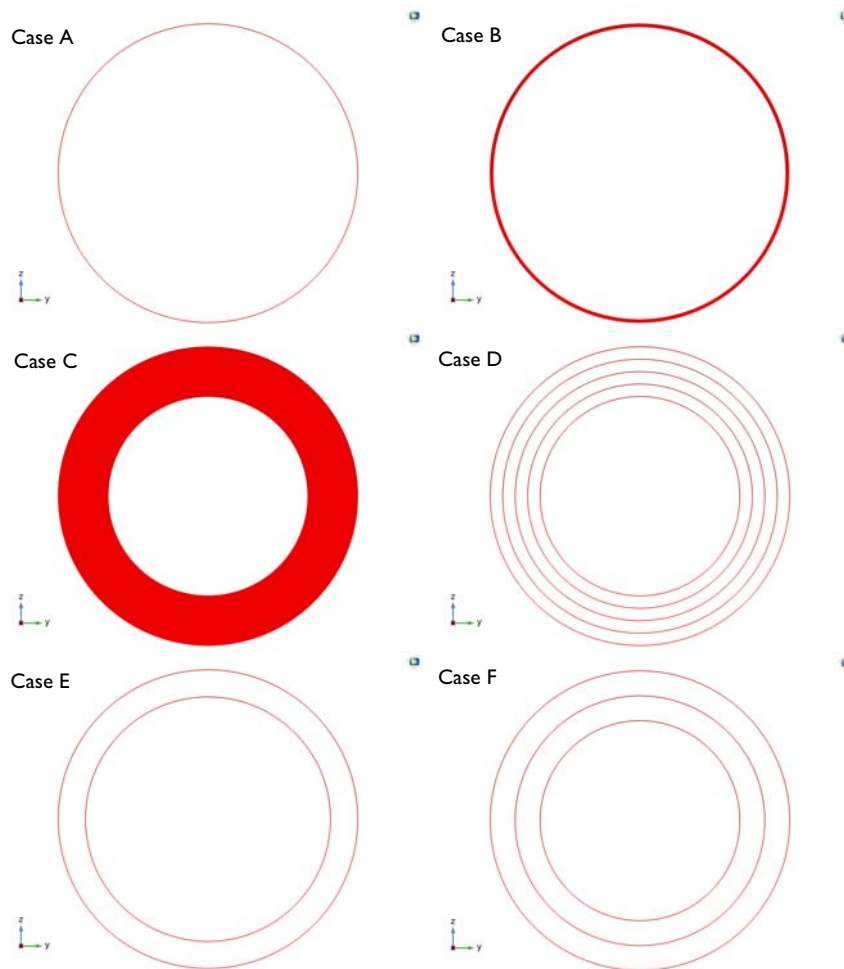


Figure 3-19: The modeled geometry and various ways to use a layered material data set in order to evaluate results in a layered material.



In the *COMSOL Multiphysics Reference Manual*: The [Layered Material](#) data set

Layered Material Slice Plot

As described in the previous section, you can generate slice plots using the layered material data set. You can however also use a **Layered Material Slice** plot. It is useful in the following cases:

- When you want to create a slice only in one or few selected layers.
- When you want to create a slice plot through some or all layers but not necessarily perpendicular to the thickness direction.
- When you want to examine a particular layer in detail and want to plot the results at a particular position within the layer which is not the midplane.



In the *COMSOL Multiphysics Reference Manual*: The [Layered Material Slice](#) plot.

Through Thickness Plot

As the composite laminates are heterogeneous in nature, particularly in the thickness direction, so it is often necessary to investigate the variation of a quantity through the thickness at certain locations on the boundary.

For this, you can use a **Through Thickness** plot in which any quantity can be plotted against the laminate thickness. You can select one or more geometric points on the boundary, or optionally create cut point data sets. It is also possible to specify the point coordinates directly.

Unlike other graphs, the result quantity is plotted on the x -axis, while the thickness coordinate is plotted on the y -axis. An example of a through-thickness plot is shown in [Figure 3-20](#).

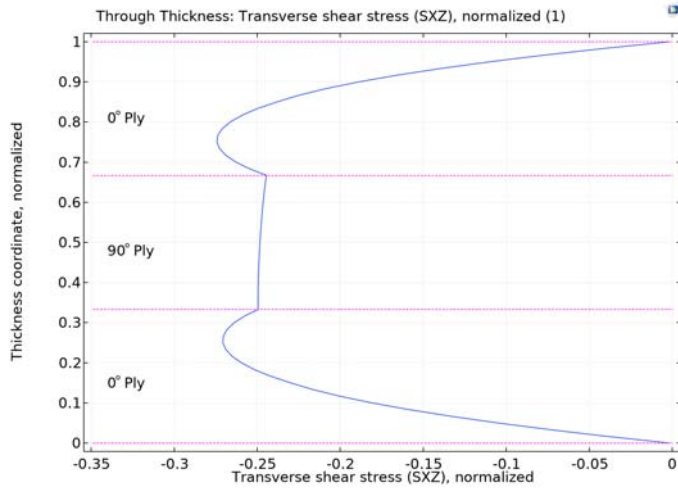


Figure 3-20: Example of a through thickness plot where the transverse shear stress is plotted at a particular point on composite laminate having three layers ([0/90/0]).



In the *COMSOL Multiphysics Reference Manual*: The [Through Thickness](#) plot.

I n d e x

- A**
 - about composite materials 10
 - about laminated composite shells 13
- B**
 - benefits of composites 12
 - buckling analysis 40
- C**
 - classification of composites 10
 - composite materials modeling 9
 - constituents of a lamina 17
- D**
 - delamination modeling 40
- E**
 - emailing COMSOL 8
 - esl theory 21
- F**
 - failure modeling 39
 - fiber arrangement 13
 - FRP composites 11
- H**
 - homogenization 18
- I**
 - internet resources 7
- K**
 - knowledge base, COMSOL 8
- L**
 - laminate coordinate system 29
 - laminated shell theories 21
 - layer selections 34
 - layered material dataset 42
 - layered material slice plot 45
 - layered material stack 31
 - layered materials 25
 - layerwise theory 23
- M**
 - macromechanics 19
 - micromechanics 17
 - multiphysics modeling 41
 - multiscale analysis 15
- P**
 - preview plots 26
- R**
 - results evaluation 42
- S**
 - saving and loading layered materials 32
 - stacking sequence 13
 - sweeping over layered materials 33
- T**
 - technical support, COMSOL 8
 - through thickness plot 45
 - types of laminate 15
- W**
 - websites, COMSOL 8

