COMSOL MULTIPHYSICS

STRUCTURAL MECHANICS MODULE

USER'S GUIDE

VERSION 3.4

COMSOL

How to contact COMSOL:

Benelux
COMSOL BV
Röntgenlaan 19
2719 DX Zoetermeer
The Netherlands
Phone: +31 (0) 79 363 4230
Fax: +31 (0) 79 361 4212
info@femlab.nl
www.femlab.nl

Denmark
COMSOL A/S
Diplomvej 376
2800 Kgs. Lyngby
Phone: +45 88 70 82 00
Fax: +45 88 70 80 90
info@comsol.dk
www.comsol.dk

Finland
COMSOL OY
Arabianranta 6
FIN-00560 Helsinki
Phone: +358 9 2510 400
Fax: +358 9 2510 4010
info@comsol.fi
www.comsol.fi

France
COMSOL France
WTC, 5 pl. Robert Schuman
F-38000 Grenoble
Phone: +33 (0)4 76 46 49 01
Fax: +33 (0)4 76 46 07 42
info@comsol.fr
www.comsol.fr

Germany
FEMLAB GmbH
Berliner Str. 4
D-37073 Göttingen
Phone: +49-551-99721-0
Fax: +49-551-99721-29
info@femlab.de
www.femlab.de

Italy
COMSOL S.r.l.
Via Vittorio Emanuele II, 22
25122 Brescia
Phone: +39-030-3793800
Fax: +39-030-3793899
info.it@comsol.com
www.it.comsol.com

Norway
COMSOL AS
Søndre gate 7
NO-7485 Trondheim
Phone: +47 73 84 24 00
Fax: +47 73 84 24 01
info@comsol.no
www.comsol.no

Sweden
COMSOL AB
Tegnérgatan 23
SE-111 40 Stockholm
Phone: +46 8 412 95 00
Fax: +46 8 412 95 10
info@comsol.se
www.comsol.se

Switzerland
FEMLAB GmbH
Technoparkstrasse 1
CH-8005 Zürich
Phone: +41 (0)44 445 2140
Fax: +41 (0)44 445 2141
info@femlab.ch
www.femlab.ch

United Kingdom
COMSOL Ltd.
UH Innovation Centre
College Lane
Hatfield
Hertfordshire AL10 9AB
Phone:+44-(0)-1707 284747
Fax: +44-(0)-1707 284746
info.uk@comsol.com
www.uk.comsol.com

United States
COMSOL, Inc.
1 New England Executive Park
Suite 350
Burlington, MA 01803
Phone: +1-781-273-3322
Fax: +1-781-273-6603

COMSOL, Inc.
10850 Wilshire Boulevard
Suite 800
Los Angeles, CA 90024
Phone: +1-310-441-4800
Fax: +1-310-441-0868

COMSOL, Inc.
744 Cowper Street
Palo Alto, CA 94301
Phone: +1-650-324-9935
Fax: +1-650-324-9936

info@comsol.com
www.comsol.com

For a complete list of international representatives, visit www.comsol.com/contact

Company home page
www.comsol.com

COMSOL user forums
www.comsol.com/support/forums

Structural Mechanics Module User's Guide

Version: October 2007 COMSOL 3.4

C O N T E N T S

Chapter 1: Introduction

Chapter 2: Overview

Chapter 3: Quick Start

Chapter 4: Structural Mechanics Modeling

Chapter 5: Application Mode Guide

Chapter 6: Coordinate Systems and Symbols

Chapter 7: Continuum Application Modes

Chapter 8: Mindlin Plates

Chapter 9: Beams

Chapter 10: Trusses

Chapter 11: Shells

Chapter 12: Piezoelectric Application Modes

Chapter 13: Predefined Multiphysics Couplings

Chapter 14: Fatigue Analysis

Chapter 15: Glossary

1

Introduction

The Structural Mechanics Module 3.4 is an optional package that extends the COMSOL Multiphysics® modeling environment with customized user interfaces and functionality optimized for structural analysis. Like all modules in the COMSOL family, it provides a library of prewritten ready-to-run models that make it quicker and easier to analyze discipline-specific problems.

This particular module solves problems in the fields of structural and solid mechanics, adding special elements such as beams, plates, and shells. It provides static, eigenfrequency, damped eigenfrequency, time-dependent, quasi-static transient, parametric, linear buckling, fatigue, and frequency response analysis capabilities. You can use both linear and nonlinear material models such as elasto-plastic models and include large deformation effects as well as contact and friction in an analysis. Material models can be isotropic, orthotropic, or fully anisotropic. Define loads, constraints, and material models in a global coordinate system or in local, user-defined coordinate systems. Piezoelectric materials can be analyzed with the constitutive relations on either stress-charge or strain-charge form.

All application modes in this module are fully multiphysics enabled, making it possible to couple them to any other physics application mode in COMSOL Multiphysics or the other modules. Coupling structural analysis with thermal analysis is one example of multiphysics easily implemented with the Structural Mechanics Module, which provides predefined multiphysics couplings for thermal-structural analysis and other types of multiphysics. Piezoelectric materials, coupling the electric field and strain in both directions are fully supported inside the module through special application modes solving for both the electric potential and displacement. Structural mechanics couplings are common in simulations done with COMSOL Multiphysics and occur in interaction with fluid flow (FSI), chemical reactions, acoustics, electric fields, magnetic fields, and optical wave propagation.

The underlying equations for structural mechanics are automatically available in all of the application modes—a feature unique to COMSOL Multiphysics. This also makes nonstandard modeling easily accessible. For example, you can change the constitutive equations to model nonlinear materials. The Structural Mechanics Module also features extensible material and beam cross-section libraries.

Further, you can include accurate finite element models as blocks in a dynamic simulation performed with Simulink. This combination reduces the need for approximations and *ad hoc* models in dynamic simulations. COMSOL Multiphysics' tight integration with the COMSOL Script and MATLAB environments makes the Structural Mechanics Module very versatile. For instance, you can use a function to describe loads and constraints.

The documentation set for The Structural Mechanics Module consists of three books. The one in your hands, the *Structural Mechanics Module User's Guide*, introduces the basic functionalities in the module, reviews new features in the version 3.4 release, reviews basic modeling techniques through tutorial and benchmark example models, and includes reference material of interest to those working in structural mechanics. The second book in the set, the *Structural Mechanics Module Model Library*, contains a large number of ready-to-run models that illustrate a broad range of applications. Each model comes with theoretical background as well as step-by-step instructions that illustrate how to set it up. Further, we supply these models as Model MPH-files so that you can open them in COMSOL Multiphysics for immediate access. This way you can follow along with the printed discussion as well as use them as a jumping-off point for your own modeling needs. A third book, the *Structural Mechanics Module Reference Guide*, contains reference material about command-line functions and programming. All documentation is available in HTML and PDF format from the COMSOL Help Desk.

Typographical Conventions

All COMSOL manuals use a set of consistent typographical conventions that should make it easy for you to follow the discussion, realize what you can expect to see on the screen, and know which data you must enter into various data-entry fields. In particular, you should be aware of these conventions:

- A **boldface** font of the shown size and style indicates that the given word(s) appear exactly that way on the COMSOL graphical user interface (for toolbar buttons in the corresponding tooltip). For instance, we often refer to the **Model Navigator**, which is the window that appears when you start a new modeling session in COMSOL; the corresponding window on the screen has the title **Model Navigator**. As another example, the instructions might say to click the **Multiphysics** button, and the boldface font indicates that you can expect to see a button with that exact label on the COMSOL user interface.
- The names of other items on the graphical user interface that do not have direct labels contain a leading uppercase letter. For instance, we often refer to the Draw toolbar; this vertical bar containing many icons appears on the left side of the user interface during geometry modeling. However, nowhere on the screen will you see the term "Draw" referring to this toolbar (if it were on the screen, we would print it in this manual as the **Draw** menu).
- The symbol **>** indicates a menu item or an item in a folder in the **Model Navigator**. For example, **Physics>Equation System>Subdomain Settings** is equivalent to: On the **Physics** menu, point to **Equation System** and then click **Subdomain Settings**. **COMSOL Multiphysics>Heat Transfer>Conduction** means: Open the **COMSOL Multiphysics** folder, open the **Heat Transfer** folder, and select **Conduction**.
- A `Code` (monospace) font indicates keyboard entries in the user interface. You might see an instruction such as "Type `1.25` in the **Current density** edit field." The monospace font also indicates COMSOL Script codes.
- An *italic* font indicates the introduction of important terminology. Expect to find an explanation in the same paragraph or in the Glossary. The names of books in the COMSOL documentation set also appear using an italic font.

2

Overview

This manual describes the Structural Mechanics Module version 3.4. It is intended to give you an introduction to the modeling stages within the Structural Mechanics Module and to provide a detailed example of how to work with the models in this set, as well as serve as a reference for more advanced modeling.

- Chapter 3, “Quick Start,”, gives you the knowledge necessary to start using COMSOL Multiphysics with the Structural Mechanics Module.
- In Chapter 4, “Structural Mechanics Modeling,” you can find modeling advice for various structural mechanics problems.
- Chapter 5, “Application Mode Guide,”; Chapter 7, “Continuum Application Modes,”; Chapter 8, “Mindlin Plates,”; Chapter 9, “Beams,”; Chapter 10, “Trusses,”; and Chapter 11, “Shells,”, provide guidelines to help you select and use the application modes in this module.
- Chapter 14, “Fatigue Analysis,”; show you have to perform fatigue analysis together with COMSOL Script or MATLAB.
- The “Application Mode Programming Reference” in Chapter 3 describes the powerful integration with COMSOL Script and MATLAB and gives details about the implementation of the application modes.

A separate book, the *Structural Mechanics Module Model Library*, describes in great detail each entry in the collection of software models you received with this product.

What Can the Structural Mechanics Module Do?

The Structural Mechanics Module contains a set of modeling interfaces, called application modes, adapted to a broad category of structural-mechanics analysis. The module serves as an excellent tool for the professional engineer, researcher, and teacher. In education the benefit of the short learning curve is especially useful because educators need not spend excessive time learning the software and can instead focus on the modeling process.

As you develop models using the Structural Mechanics Module, you can view them in terms of the underlying partial differential equations or the principle of virtual work. The software thus offers a unique way to understand the laws of physics and the equations that describe them. For instance, you can add additional physics like viscoelasticity to the constitutive equations. It is also possible to export a simulation into the COMSOL Script or MATLAB environments, or save it as a Model M-file script. This makes it possible to incorporate COMSOL Multiphysics models with products in the MATLAB family such as Simulink and the Control System Toolbox.

What Problems Can It Solve?

The Structural Mechanics Module is a collection of application modes for COMSOL Multiphysics that handles static, eigenfrequency and damped eigenfrequency, transient, frequency response, quasi-static, linear buckling, and parametric analyses for applications in structural mechanics, solid mechanics, and piezoelectricity.

STATIC ANALYSIS

In a static analysis the load and constraints are fixed in time.

EIGENFREQUENCY ANALYSIS

An eigenfrequency analysis finds the undamped eigenfrequencies and mode shapes of a model. Sometimes referred to as the free vibration of a structure.

DAMPED EIGENFREQUENCY ANALYSIS

A damped eigenfrequency analysis finds the damped eigenfrequencies and mode shapes of a model. The quality and decay factors of the structure are also calculated.

TRANSIENT ANALYSIS

A transient analysis finds the transient response for a time-dependent model, taking into account mass, mass moment of inertia, and damping.

FREQUENCY RESPONSE ANALYSIS

A frequency-response analysis finds the steady-state response from harmonic loads.

QUASI-STATIC ANALYSIS

A quasi-static analysis neglects mass effects, assuming the time scale in the structural-mechanics problem is much smaller than other dynamics. An example is a transient thermal problem where the time scale in the thermal problem is often much longer than the one in the structural dynamics.

PARAMETRIC ANALYSIS

A parametric analysis finds the solution dependence due to the variation of a specific parameter, which could be, for instance, a material property or the position of a load.

LARGE DEFORMATIONS

You can include large deformations with the restriction of small strains in all fully dimensional application modes. This effect is also sometimes referred to as a nonlinear geometric effect. Using large deformations, the normal strain is replaced with the Green strain, and the stress is replaced with the second Piola-Kirchhoff stress. COMSOL Multiphysics solves the problem using a *total Lagrangian formulation*. Large deformation is only available in the continuum application modes.

LINEAR BUCKLING ANALYSIS

A linear buckling analysis includes the stiffening effects from stresses coming from nonlinear strain terms. The two stiffnesses coming from stresses and material define an eigenvalue problem where the eigenvalue is a load factor that, when multiplied with the actual load, gives the critical load in a linear context.

Linear buckling analysis uses the eigenvalue solver.

Another way to calculate the critical load is to include large deformation effects and increase the load until the solver fails because the load has reached its critical value.

Linear buckling analysis is only available in the continuum application modes.

ELASTO-PLASTIC ANALYSIS

An elasto-plastic analysis involves a nonlinear material model with or without hardening. Two different hardening models are available:

- Isotropic
- Kinematic

Elasto-plastic analysis is available in the continuum application modes.

FATIGUE ANALYSIS

A fatigue analysis is done in order to find the fatigue damage or fatigue life of a component. Fatigue analysis is divided into high-cycle and low-cycle fatigue depending of the number of load cycles. The Structural Mechanics Module as delivered can handle the following cases for both high-cycle and low-cycle fatigue:

- Proportional loading constant amplitude
- Nonproportional loading constant amplitude
- Proportional loading nonconstant amplitude

Note: The fatigue analysis in the Structural Mechanics Module is script based and requires COMSOL Script or MATLAB.

APPLICATIONS

Examples of applications include thin plates loaded in a plane (plane stress), thick structures with no strain in the out-of-plane direction (plane strain), axisymmetric structures, frame structures in 2D and 3D, thin-walled 3D structures (shells), and general 3D structures modeled using solid elements.

Available application modes are:

- Plane stress
- Plane strain
- Axial symmetry, stress-strain
- 2D beams, Euler theory
- 2D truss
- Thick plates, Mindlin theory
- 3D beams, Euler theory

- 3D truss
- Solid, stress-strain
- Shells
- Piezoelectric application modes
  - Piezo solid
  - Piezo plane stress
  - Piezo plane strain
  - Piezo axial symmetry

New Features in Structural Mechanics Module 3.4

- Improved piezoelectric application modes. See “Piezoelectric Application Modes” on page 319 for more information.
- Follower loads. See “Follower Loads” on page 73 for more information.
- Fatigue analysis capabilities. See “Fatigue Analysis” on page 125 for more information.

3

Quick Start

The objective of this chapter is to familiarize you with modeling procedures in the Structural Mechanics Module using the graphical user interface. Because this module is fully integrated with COMSOL Multiphysics, the modeling process is similar to the one used in that environment. This chapter takes a detailed walk-through of one model to present the various aspects of the simulation process; it steps through all the stages of modeling, from geometry creation to postprocessing.

Basic Modeling Procedures

The way COMSOL Multiphysics orders its toolbar buttons and menus mirrors the basic procedural flow during a modeling session. You work your way from left to right in the process of modeling, defining, solving, and postprocessing a problem using the COMSOL Multiphysics graphical user interface (GUI). Thus, in this manual and the accompanying *Structural Mechanics Model Library* manual we maintain a certain style convention when describing the following introductory models as well as those in the *COMSOL Multiphysics Model Library.* The format includes headlines that corresponding to each major step in the modeling process; these headlines also roughly correspond to the various GUI modes and menus.

Model Navigator

The **Model Navigator** appears when you start COMSOL Multiphysics or when you restart from scratch within COMSOL Multiphysics by selecting **New** from the **File** menu or clicking on the **New** button on the Main toolbar. On the **New** page in the **Model Navigator** you specify the application mode, names of dependent variables, and the nature of the problem—the analysis type: stationary (static), eigenfrequency or damped eigenfrequency, time dependent (transient), quasi-static, parametric, or frequency response. By clicking on the **Multiphysics** button you can set up a combination of application modes from the Structural Mechanics Module, other modules, or COMSOL Multiphysics. It is also possible to open the **Model Navigator** from the **Multiphysics** menu at any time to add or remove an application mode.

Options and Settings

This section reviews basic settings, for example, those for the axes and grid spacing. All settings are accessible from the **Options** menu and some are also accessible by double-clicking the status bar. You might need to use the **Constants** dialog box to enter model parameters (see “Constants” on page 139 in the *COMSOL Multiphysics User's Guide* for a more detailed description). You can also maintain libraries of user-defined materials, which you access through the **Materials/Coefficients Library** dialog box.

It is possible to define materials, loads, and constraints in a user-defined coordinate system. Create such a coordinate system in the **Coordinate System Settings** dialog box, which is accessible from the **Options** menu.

Geometry Modeling

In this step you set up the model's geometry. This stage requires knowledge of how to use the **Draw** menu and the Draw toolbar (see "Geometry Modeling and CAD Tools" on page 23 in the *COMSOL Multiphysics User's Guide* for details). You can also import 3D CAD drawings using the optional CAD Import Module.

Physics Settings

In this section you define a model's physics. Open all appropriate dialog boxes from the **Physics** menu or by double-clicking on the domain in the respective domain selection mode. Further, control the active selection mode from the **Physics** menu or by clicking on the appropriate domain type button on the Main toolbar.

Material properties are normally defined on subdomains. In the Structural Mechanics Module, however, some application modes are not defined at the subdomain level. This is the case, for example, in

- The Shell application mode, where you define material properties on boundaries (3D) / faces
- The In-plane Euler Beam application mode, where you define material properties on boundaries (2D) / edges.

APPLICATION MODE PROPERTIES

Application mode properties are global properties controlling the analysis starting with which analysis to perform. Make all corresponding settings in the **Application Mode Properties** dialog box, which you open by choosing **Physics>Properties**.

APPLICATION SCALAR VARIABLES

Application scalar variables are global variables defining an analysis. The Structural Mechanics Module application modes have only one variable, `freq`, the excitation frequency in a frequency response analysis. To open the **Application Scalar Variables** dialog box, choose **Scalar Variables** from the **Physics** menu.

POINT SETTINGS

You define loads and constraints on points in Point mode. Point settings are used in all Structural Mechanics Module application modes, whereas point masses are defined in the Beam application modes. A table is used to describe the settings in the **Point Settings** dialog box in a compact format.

EDGE SETTINGS

Edge settings are used only in the 3D application modes for shells, solids, and beams. In the 3D Euler Beam application mode you define physical properties as well as loads and constraints, whereas the other application modes define only loads and constraints. A table is used to describe the settings in the **Edge Settings** dialog box in a compact format.

BOUNDARY SETTINGS

In Boundary Selection mode, you specify loads and constraints on the edges (in 2D) or faces (in 3D). For in-plane Euler beams and shells the physical properties are also defined. A table is used to describe the settings in the **Boundary Settings** dialog box in a compact format.

SUBDOMAIN SETTINGS

In Subdomain Selection mode you specify material properties, loads, and damping for application modes existing on the top domain in 2D and 3D. A table is used to describe the settings in the **Subdomain Settings** dialog box in a compact format. You can also implement constraints, specify initial stresses and strains, and control the element to use from this dialog box. For all continuum application modes you can select the order of the Lagrange element. In addition, for time-dependent and nonlinear problems you set initial conditions for subdomains in this mode.

Mesh Generation

At this stage the software meshes the problem geometry. Sometimes you simply click one of the meshing buttons on the Main toolbar; in other cases it is necessary to set some parameters using the **Mesh** menu and the dialog boxes for the meshers, or use the interactive meshing and the Mesh toolbar (see "Meshing" on page 285 in the *COMSOL Multiphysics User's Guide* for a detailed description about meshing).

Computing the Solution

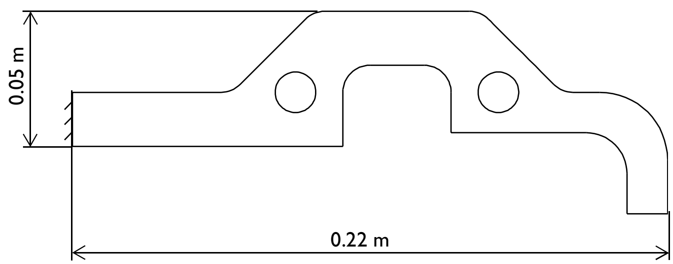
To compute the solution, click the **Solve** button on the Main toolbar. You can specific settings for each solver in the **Solver Parameters** dialog box (see "Solving the Model" on page 359 in the *COMSOL Multiphysics User's Guide* for details).

Postprocessing and Visualization

COMSOL Multiphysics' powerful visualization tools are accessible in the program's Postprocessing mode, but to use them you must be familiar with the **Postprocessing** menu (see "Postprocessing and Visualization" on page 419 in the *COMSOL Multiphysics User's Guide* for details). For additional postprocessing you can export the solution to the COMSOL Script or MATLAB workspace.

A Mechanical Component

The following detailed example consists of a plane stress analysis of a mechanical component. The component is cut from a 4-mm steel plate, and the thickness is small compared to its extent in the *xy*-plane. This aspect along with the fact that the applied loads lie in the *xy*-plane mean that you can assume that the out-of-plane stress is negligible. The Plane Stress application mode works under this assumption.



This example also introduces the seven basic analysis types available in the Structural Mechanics Module:

- Static analysis
- Eigenfrequency analysis
- Damped eigenfrequency analysis
- Time-dependent analysis
- Frequency response analysis
- Parametric analysis
- Quasi-static transient analysis

In the static, time-dependent, frequency response, parametric, and quasi-static analyses, the model contains a load to the lower-right end of the component. In all analyses the component is clamped on the left-hand side.

Static Analysis

A static analysis has no explicit or implicit time dependencies. This situation corresponds to the steady state of a transient analysis with constant boundary conditions and material properties.

The purpose of this analysis is to:

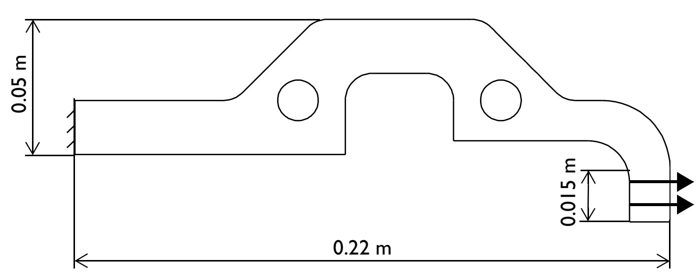
- Find the maximum stress level and compare it with the material's yield strength
- Find the static deflection at the point where the load is applied and compare it with the time-dependent analysis.

The Model Library note immediately below appears in the discussion of every model. The path indicates the location of the model file on the **Model Library** page in the **Model Navigator**.

Model Library path: `Structural_Mechanics_Module/Tutorial_Models/component_static`

MODEL DEFINITION

The model starts with a mechanical component whose shape and overall dimensions are shown in the following figure. It is possible to create the geometry in a CAD package and import it into COMSOL Multiphysics as a DXF file.



Some key parameters for the model:

Material

- Structural steel as taken from the material library
- Thickness of 4 mm

Load

A 900 N force in the x direction on the inside of the right end

Constraints

The left edge is fixed.

RESULTS

After the analysis you find that the von Mises effective stress has a maximum value of $2.8 \cdot 10^8$ N/m^2 = 280 MPa, which, compared with the material's yield strength of 350 MPa, results in a utilization factor of 80%.

The analysis also gives the static displacements at the bottom end of the edge where the load is applied:

RESULT	x DIRECTION	y DIRECTION
Displacement	6.89e-4 m	1.14e-3 m

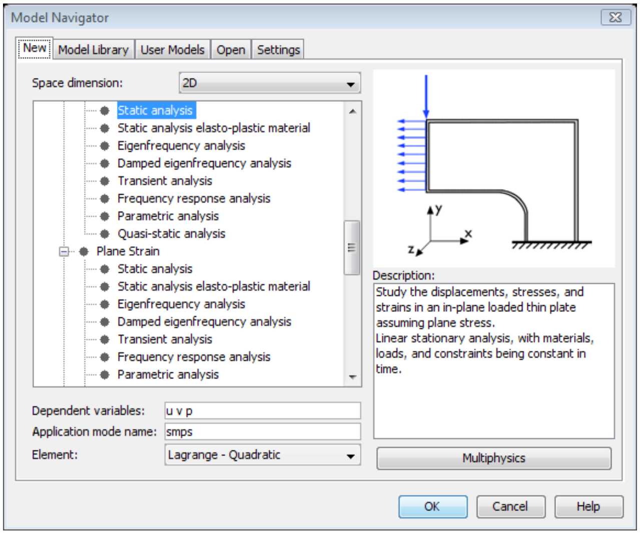
Now take a step-by-step look at how to achieve these results using the Structural Mechanics Module.

MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

1 In the **Model Navigator** go to the **New** page, then select **2D** from the **Space dimension** list.

2 On that same page go to the list of application modes and select **Structural Mechanics Module>Plane Stress>Static analysis**.



3 Click **OK** to close the **Model Navigator**.

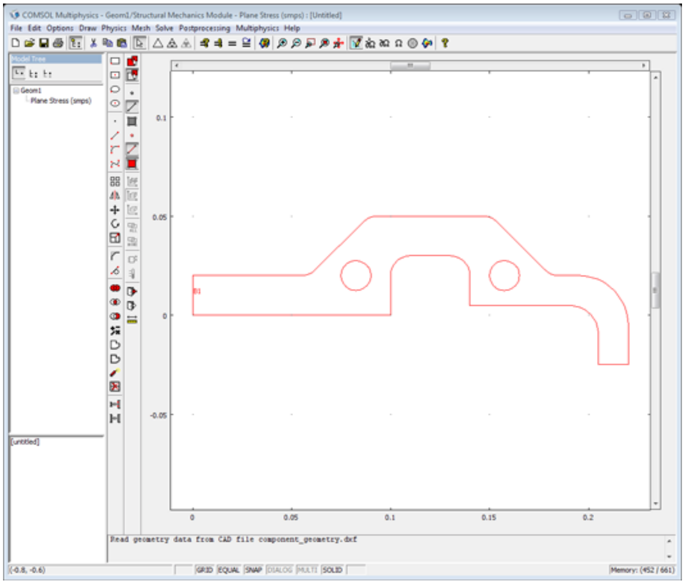
Geometry Modeling

Instead of drawing the geometry directly in the user interface, you can import a DXF file.

1 From the **File** menu select **Import>CAD Data From File**.

2 The **Import CAD Data from File** dialog box appears. Open the DXF file `component_geometry.dxf`, which is located in `/models/Structural_Mechanics_Module/Tutorial_Models` in the COMSOL installation directory.

3 Click **Import** to close the dialog box, then click the **Zoom Extents** button on the Main toolbar to view the entire geometry.

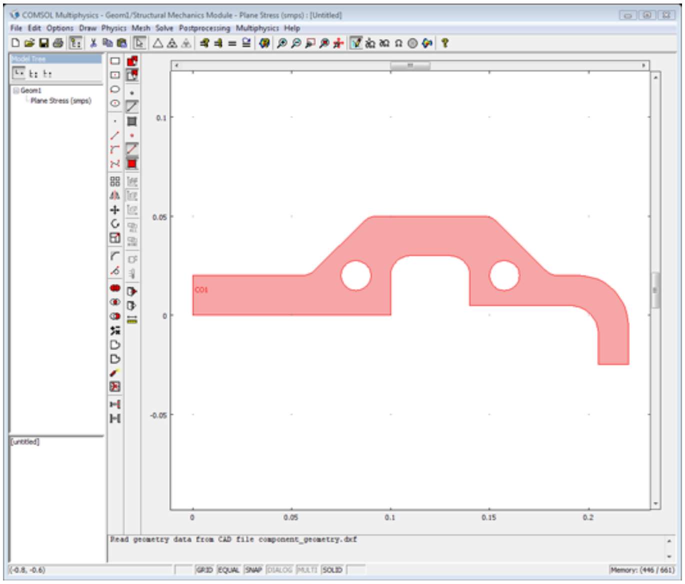


4 Select all the lines in the drawing by clicking the left mouse button and dragging a rubber-band box that encloses all the edges (or use the shortcut key Ctrl+A).

5 Construct a solid from the edges imported from the DXF file. To coerce the edges to a solid, click the **Coerce to Solid** button in the Draw toolbar.

6 Split the solid by clicking the **Split Object** button in the Draw toolbar to create independent objects for the holes.

7 Click the **Difference** button on the Draw toolbar to cut out the hole in the geometry.



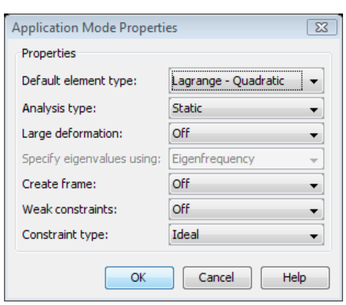
Physics Settings

Application Mode Properties

Set the analysis type to static (the analysis type is already static in this model, so this step is not necessary).

1 From the **Physics** menu select **Properties** to open the **Application Mode Properties** dialog box.

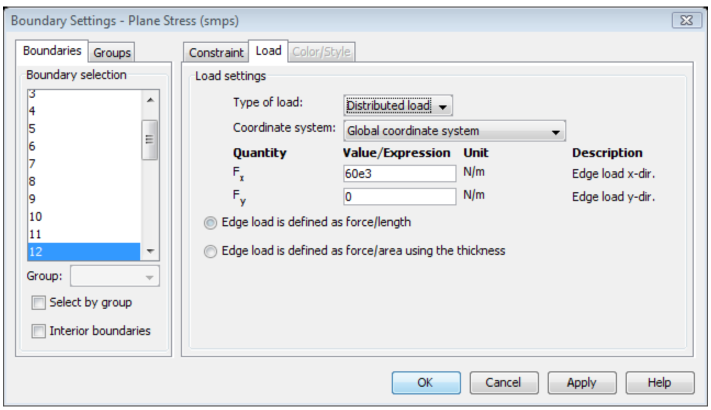
2 The **Analysis type** list defines which analysis to perform and some other properties for the analysis. Because you selected a static analysis from the **Model Navigator**, the analysis type is already defined as being **Static**.



Boundary Settings In boundary mode you specify loads and constraints. By default all edges are free, which means that there are no loads or constraints. Loads can be defined as force per area (using the thickness) or as force per length. The default is force per length, which this example uses.

The total static force on Edge 12 on the lower right end is 900 N. This results in a distributed force along the edge, which is 15 millimeter long, of 60 kN/meter. The left edge is fixed in both directions.

1 Open the **Boundary Settings** dialog box by going to the **Physics** menu and selecting **Boundary Settings**.
2 Select Boundary 1. Click the **Constraint** tab, then select **Fixed** from the **Constraint condition** list.
3 Select Boundary 12. Click the **Load** tab and enter `60e3` in the $\mathbf{F_x}$ edit field.



You have set the boundary conditions according to the following table:

	BOUNDARY 1		BOUNDARY 12	
Page	Constraint		Load	
	Constraint condition	Fixed	F_x	60e3

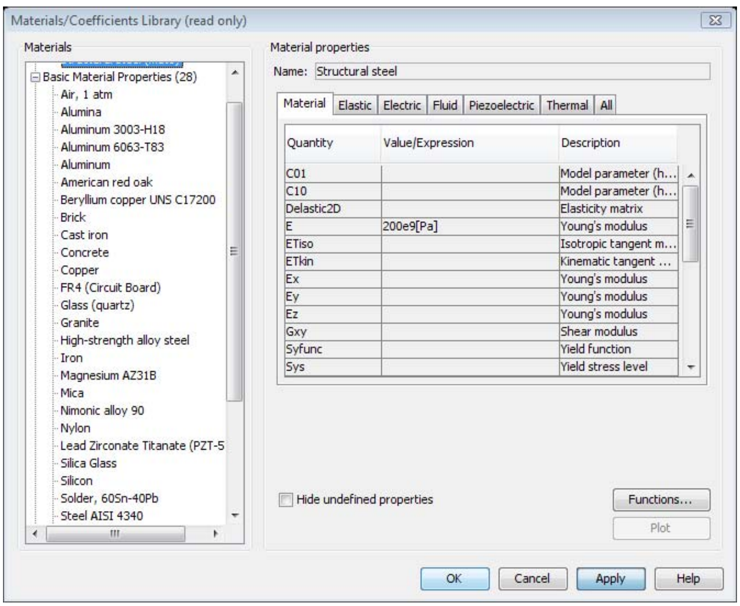
You frequently encounter tables such as this one both in the remainder of this chapter and throughout the Model Library. The row marked “Page” indicates on which page in the dialog box you find the setting.

4 Click **OK** to close the **Boundary Settings** dialog box.

Subdomain Settings In subdomain mode you specify material properties and element order. The material properties are selected from the material library or entered explicitly by typing the corresponding value or expression. This example uses the material library. Before you can use a material from the library, you must add it to the geometry. Do so from the **Options** menu or directly from the **Subdomain Settings** dialog box.

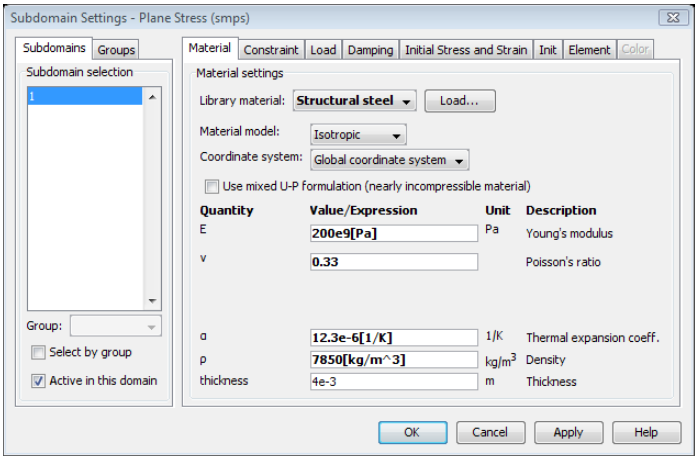
1 Select **Subdomain Settings** from the **Physics** menu.
2 Select Subdomain 1.
3 Click the **Load** button on the **Material** page to open the **Materials/Coefficients Library** dialog box.
4 Select **Structural steel** from the **Basic Material Properties** folder in the **Materials** list and click **Apply**.

   You have now added the structural steel material to the geometry, an entry that you can see in the **Model** part of the **Materials** list.



5 Click **OK** to close the **Material/Coefficients Library** dialog box. **Structural steel** is now selected in the **Library material** list in the **Subdomain Settings** dialog box.

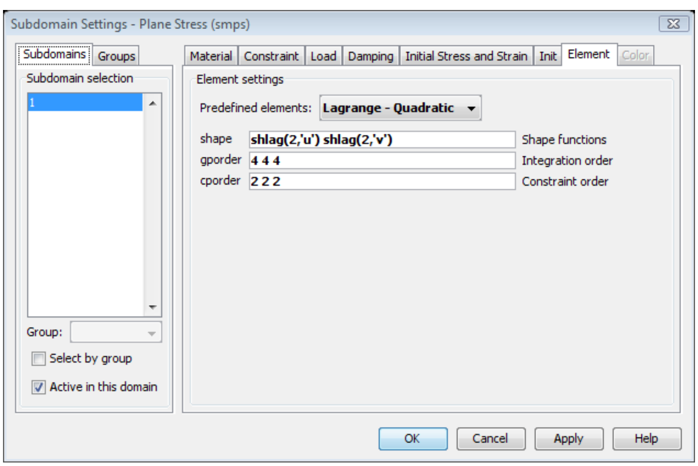
6 Type `4e-3` in the **thickness** edit field.



In tabular form, the material settings are:

	SUBDOMAIN 1	
Page	Material	
	Library material	Structural steel
	thickness	`4e-3`

The **Element** page shows the shape functions currently in use in the selected subdomain. The shape functions define polynomials used in interpolating the dependent variables. Typically, you do not need to change this setting.

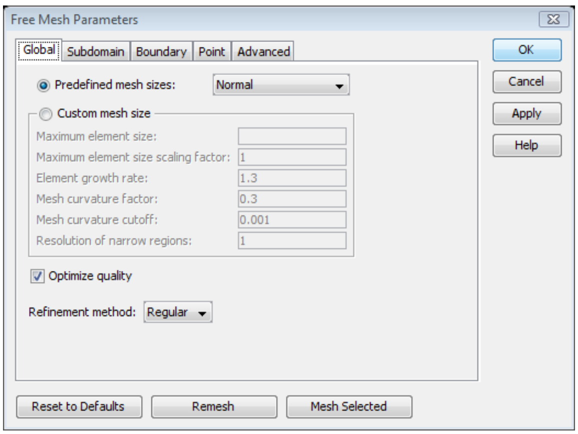


The default element type is quadratic Lagrange elements. They use 2nd-order polynomials, which is often a good trade-off between memory usage and accuracy. You can use linear Lagrange elements to reduce memory consumption when accurate stress or strain results are not required.

Mesh Generation

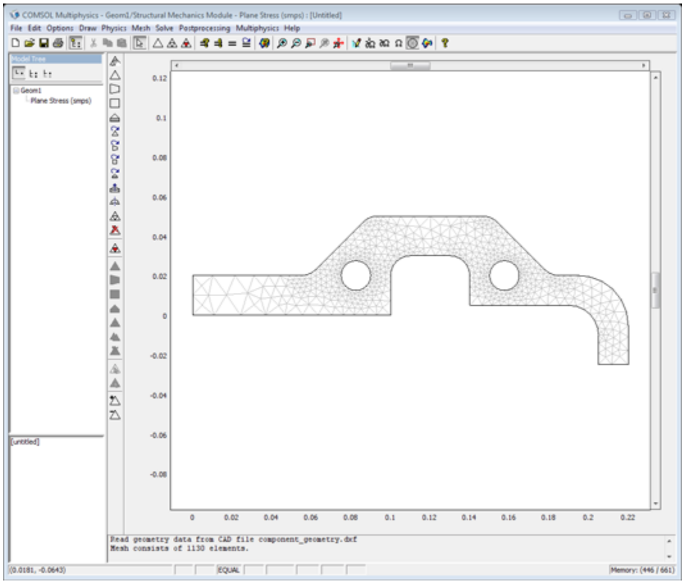
The **Free Mesh Parameters** dialog box, found on the **Mesh** menu, gives access to a number of parameters that control the mesh's density and gradation. Use the default mesh parameters.

1 Select **Free Mesh Parameters** from the **Mesh** menu to look at the **Free Mesh Parameters** dialog box.



2 Click **OK** to use the default mesh settings.

3 Initialize the mesh by clicking the **Initialize Mesh** button on the Main toolbar.



Computing the Solution

The analysis type controls which solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. This option is enabled as the default, so there is no need to change the solver settings because stationary is the solver associated with the static analysis type.

To compute the solution, either click the **Solve** button (**=**) on the Main toolbar or select **Solve Problem** from the **Solve** menu.

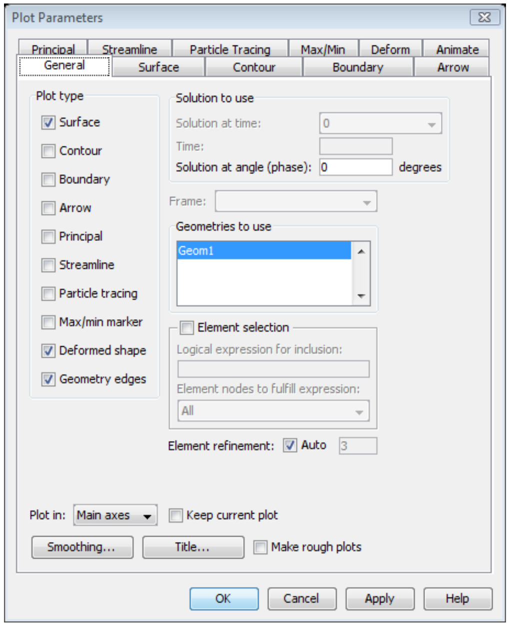
Postprocessing and Visualization

In *Postprocessing mode* you can, for example, add additional plot types and set parameters for plots. The postprocessing utilities can visualize any expression containing, for example, the solution variables, their derivatives, and the space coordinates. Many frequently used expressions are predefined as *postprocessing variables*, and they are directly available from lists in the **Plot Parameters** dialog box.

As soon as the solution is ready, a default plot appears. In the Plane Stress application mode the default visualizes the *von Mises effective stress*. Plot the *von Mises* stress together with the deformed shape of the component:

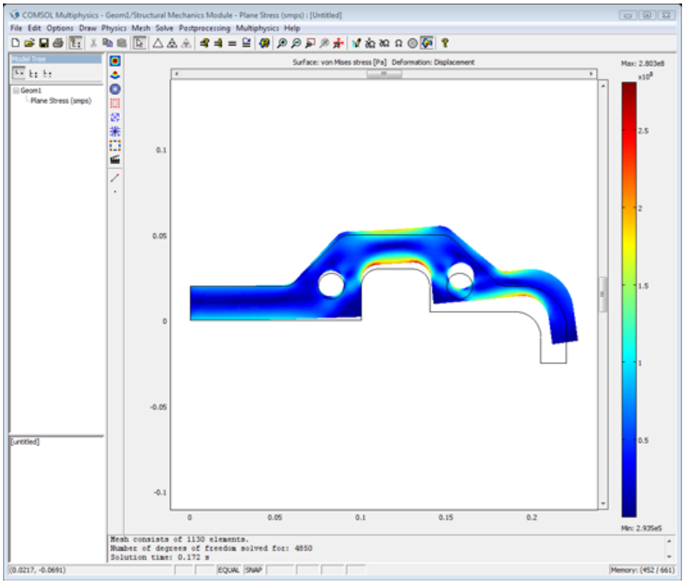
1 Select **Plot Parameters** from the **Postprocessing** menu.

2 On the **General** page select the **Deformed shape** and **Surface** check boxes.



3 Click **OK** to close the **Plot Parameters** dialog box.

4 Click the **Zoom Extents** button on the Main toolbar.



The deformation is exaggerated using automatic scaling. You can control the scaling of the deformation from the **Plot Parameters** dialog box on the **Deform** page.

Eigenfrequency Analysis

An eigenfrequency analysis finds the eigenfrequencies and modes of deformation of a component. The eigenfrequencies f in the structural mechanics field are related to the eigenvalues λ returned by the solvers through

$$f = -\frac{\mathrm{Im}(\lambda)}{2\pi}$$

In COMSOL Multiphysics you can choose between working with eigenfrequencies and working with eigenvalues according to your preferences. Eigenfrequencies are the default option for all application modes in the Structural Mechanics Module.

The purpose of the following eigenfrequency analysis is to find the six lowest eigenfrequencies and corresponding mode shapes.

Model Library path:
`Structural_Mechanics_Module/Tutorial_Models/component_eigen`

MODEL DEFINITION

The geometry, material, loads, and constraints are the same as for the static analysis; see the description on page 17 for details.

RESULTS

The first six eigenfrequencies are:

EIGENFREQUENCY NUMBER	FREQUENCY
f_1	300.67 Hz
f_2	1346.42 Hz
f_3	3456.12 Hz
f_4	4405.38 Hz
f_5	8410.73 Hz
f_6	11 kHz

MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

The eigenfrequency analysis is described as if it were done after the static analysis described on page 17, so the Plane Stress application mode is already selected.

Geometry Modeling

This was already done in the static analysis.

Physics Settings

This model uses the same material, loads, and constraints as the static analysis.

Application Mode Parameters

Change the analysis type to eigenfrequency analysis:

1 From the **Physics** menu, choose **Properties** to open the **Application Mode Properties** dialog box.

   The **Analysis type** list defines which analysis to perform and which equation to solve.

2 Select **Eigenfrequency** from the **Analysis type** list; then click **OK**.

Mesh Generation

This model uses the same mesh, so there is no difference from the static analysis.

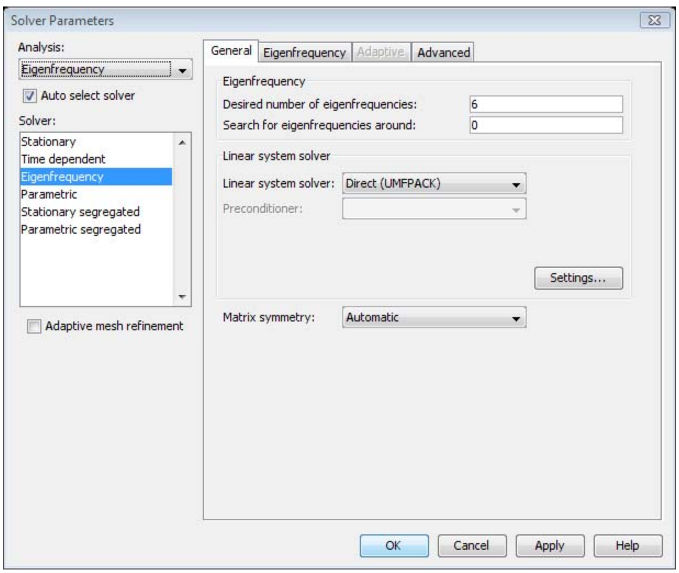
Computing the Solution

The analysis type controls which solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. This option is enabled as the default, so there is no need to change the solver settings. The eigenfrequency solver is the solver associated with the eigenfrequency analysis type.

Examine the eigenfrequencies solver parameters

1 From the **Solve** menu, choose **Solver Parameters** to open the **Solver Parameters** dialog box.

   The **Eigenfrequency** solver is already selected through the **Auto select solver** option. The number of eigenfrequencies to compute is controlled from the **General** page. Use the default settings to solve for the six lowest eigenfrequencies.



2 Click **OK** to close the **Solver Parameters** dialog box.

3 To compute the solution, either click the **Solve** button (**=**) on the Main toolbar or select **Solve Problem** from the **Solve** menu.

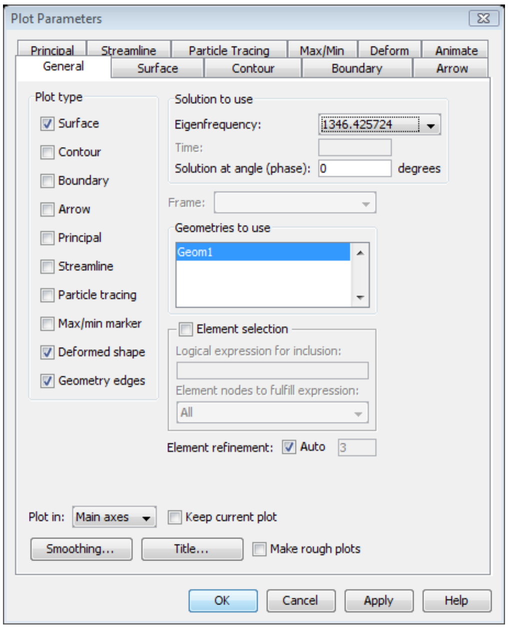
Postprocessing and Visualization

You can select which eigenfrequency to work with from a list on the **General** page in the **Plot Parameters** dialog box. The default plot shows the eigenmode corresponding to the lowest eigenfrequency. The eigenmodes are scaled (normalized) so the size of deformations should be compared only within an eigenmode and not among modes.

1 Choose **Plot Parameters** from the **Postprocessing** menu.

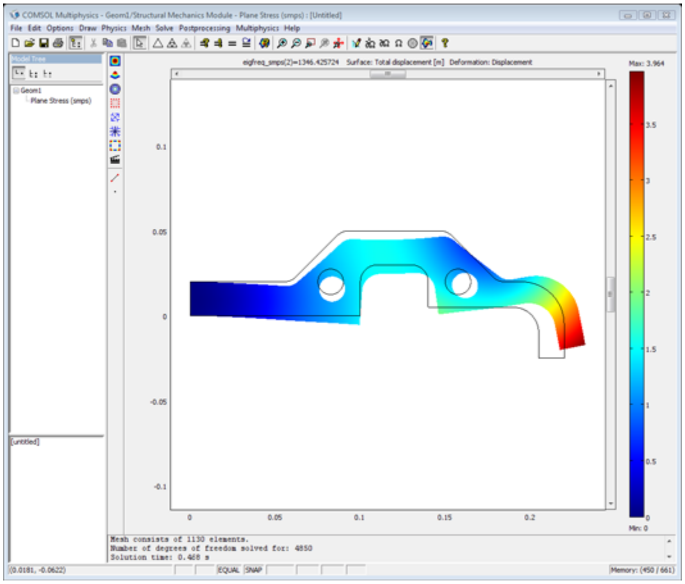
2 Select the **Deformed shape** check box on the **General** page.

3 Select the second eigenfrequency from the **Eigenfrequency** list.



4 Click the **Surface** tab and select **Total displacement** from the **Predefined quantities** list.

5 Click **OK** to close the **Plot Parameters** dialog box to look at the mode shape of the second eiegenfrequency.



Time-Dependent Analysis

A time-dependent analysis solves for the transient solution of the displacements and velocities as functions of time. The material properties, forces, and boundary conditions can vary in time.

The purpose of this analysis is to find the transient response from a harmonic load with the same amplitude as the static load during the first two periods. The excitation frequency is 500 Hz, which is between the first and second eigenfrequency found in the eigenfrequency analysis.

Model Library path: `Structural_Mechanics_Module/Tutorial_Models/component_transient`

MODEL DEFINITION

The geometry, material, and constraints are the same as for the static analysis (see the static description on page 17 for details).

Load

This model uses a harmonic load with an excitation frequency of 500 Hz on the same edge and with the same amplitude as the static problem. The expression for the load can be written

$$F_x = 900 \cdot \sin(2\pi \cdot 500 \cdot t)$$

where t denotes the time.

Damping

Damping is important in a transient analysis but can be difficult to model. For transient analysis, the Structural Mechanics Module supports Rayleigh damping and loss factor damping. It is also possible to use no damping.

This model uses Rayleigh damping, where you specify damping parameters that are proportional to the mass (α_{dM}) and stiffness (β_{dK}) in the following way:

$$C = \alpha_{dM}M + \beta_{dK}K$$

where C is the damping matrix, M is the mass matrix, and K is the stiffness matrix. The damping is specified locally; you can specify different damping parameters in different parts of the model.

You leave the damping parameters at their default values from the previous analyses because they are used only for transient and frequency response analyses.

To find good values for the Rayleigh damping, you can use the relations between the critical damping ratio and the Rayleigh damping parameters. It is often easier to interpret the critical damping ratios, which are given by

$$\xi_i = \frac{\left(\frac{\alpha_{dM}}{\omega_i} + \beta_{dK} \cdot \omega_i\right)}{2}$$

where ξ_i is the critical damping ratio at a specific angular frequency ω_i. Knowing two pairs of corresponding ξ_i and ω_i results in a system of equations

$$\begin{bmatrix} \frac{1}{(2 \cdot \omega_1)} & \frac{\omega_1}{2} \\ \frac{1}{(2 \cdot \omega_2)} & \frac{\omega_2}{2} \end{bmatrix} \begin{bmatrix} \alpha_{dM} \\ \beta_{dK} \end{bmatrix} = \begin{bmatrix} \xi_1 \\ \xi_2 \end{bmatrix}$$

with the damping parameters as the unknown variables.

The structure has a constant damping ratio of 0.1. Select two frequencies near the excitation frequency, 200 Hz and 600 Hz, to calculate the damping parameters. You can do this in COMSOL Script or MATLAB with the following commands:

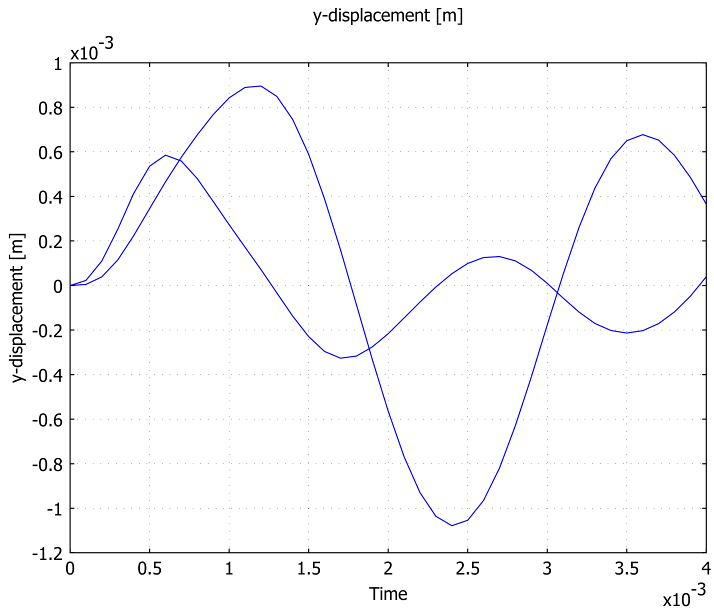
```
b=[0.1;0.1];
A=[1/(2*200*2*pi) 2*pi*200/2; 1/(2*600*2*pi) 2*pi*600/2];
% A*damp=b
damp=A\b;
alphadM=damp(1)
betadK=damp(2)
```

The result is α_{dM} = 1.88e2, β_{dK} = 3.98e-5.

For more information see the section “Damping” on page 121.

RESULTS

The following plot shows the x- and y-displacements at the bottom end of the edge where the load is applied as a function of time:



MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

The time-dependent analysis is described as if it were done after the static analysis described on page 17, so the Plane Stress application mode is already selected.

Geometry Modeling

This was already done in the static analysis (see the description on page 17 for details).

Physics Settings

Application Mode Parameters

Change the analysis type to time-dependent analysis:

1 Select **Properties** from the **Physics** menu to open the **Application Mode Properties** dialog box.

   The **Analysis type** list defines which analysis to perform and which equation to solve.

2 Select **Time dependent** from the **Analysis type** list.

3 Click **OK**.

Boundary Settings

The force at the right end is harmonic with the same amplitude as the force in the static case and with a frequency of 500 Hz,

$$F_x = 60 \cdot 10^3 \sin(2\pi 500 t)$$

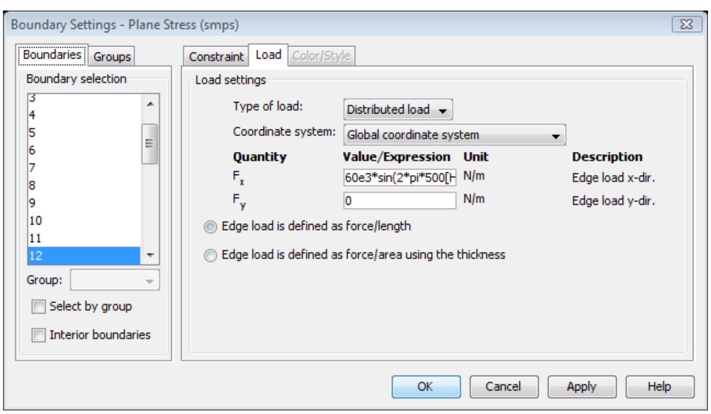
Use the unit syntax in COMSOL Multiphysics to specify the force in kN/m and the frequency in Hz. The variable t is used by COMSOL Multiphysics to denote the time in second. You specify the time steps in the **Solver Parameters** dialog box, which will be explained later in this model.

The left edge is still fixed in both directions.

1 Open the **Boundary Settings** dialog box by selecting **Boundary Settings** from the **Physics** menu.

2 Set boundary conditions according to the following table and then click **OK**:

	BOUNDARY 1		BOUNDARY 12	
Page	Constraint		Load	
	Constraint condition	Fixed	F_x	`60[kN]*sin(2*pi*500[Hz]*t)`
			F_y	0



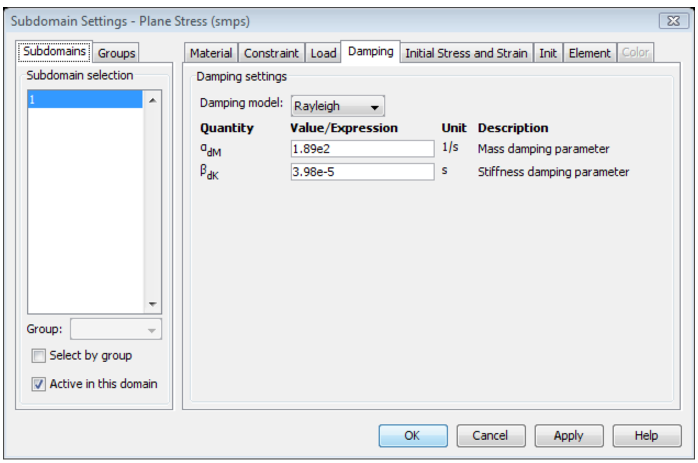
Subdomain Settings

The material properties are the same as in the static and eigenfrequency analyses.

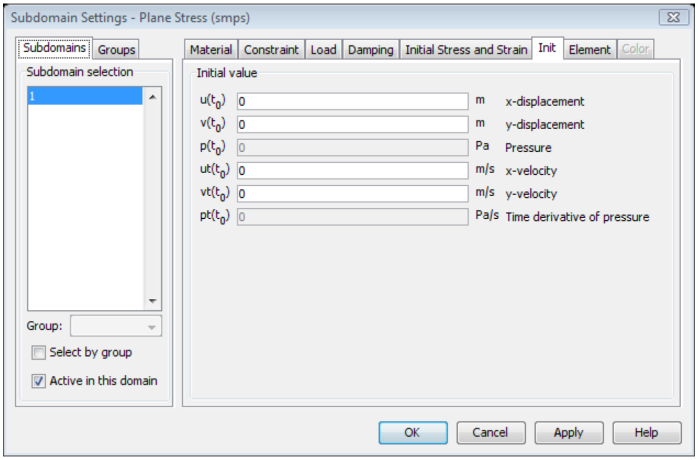
1 Select **Subdomain Settings** from the **Physics** menu.

2 Click the **Damping** tab and enter the damping properties according to the following table; when done, click **OK**.

SETTINGS	SUBDOMAIN 1
Page	Damping
α_{dM}	1.89e2
β_{dK}	3.98e-5



To specify the initial values, click the **Init** tab.



In this case the initial deformation and velocity are zero, which are the default values.

Computing the Solution

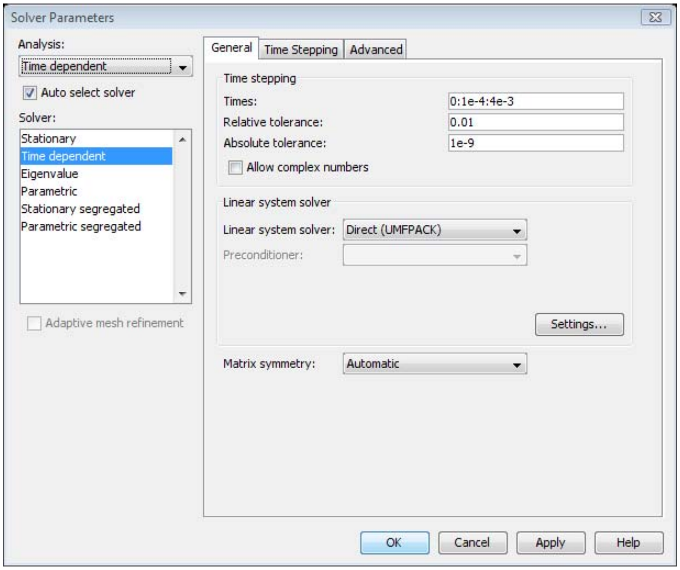
The analysis type controls which solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. The option is enabled as the default, so there is no need to change the solver settings. The time-dependent solver is the one associated with the time-dependent analysis type. Solving for two periods with an excitation frequency of 500 Hz means solving for 4 ms.

Specify the time-dependent solver parameters:

1 Choose **Solver Parameters** from the **Solve** menu.

2 Type `0:1e-4:4e-3` in the **Times** edit field. This means that the solution is saved every 0.1 ms during the total solution time of 4 ms.

3 Type `1e-9` in the **Absolute tolerance** edit field. This is important for an accurate analysis (the absolute tolerance must be smaller than the displacements).

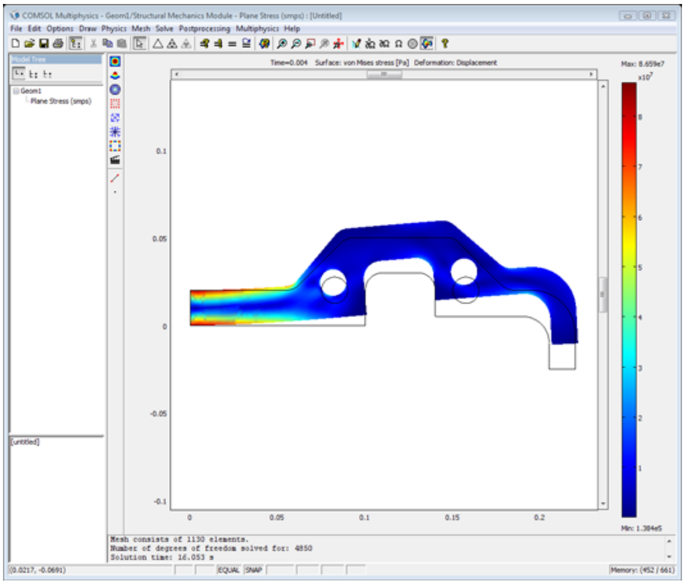


4 Click **OK**.

5 Compute the solution by clicking the **Solve** button (**=**) on the Main toolbar or by selecting **Solve Problem** from the **Solve** menu.

Postprocessing and Visualization

You have access to the solution at all the computed time steps by going to the **Plot Parameters** dialog box, then the **General** page, and selecting them from the **Output time** list. The default plot shows the last time step.



For a more quantitative view of the time evolution of the displacement, plot a graph of the displacements in the *x* and *y* directions at the lower-left corner as a function of time.

1 Select **Domain Plot Parameters** from the **Postprocessing** menu.

2 Go to the **General** page and select all the time steps from the **Solutions to use** list.

3 Click the **Point** tab (this selects a point plot automatically).

4 Click at the lower-left corner or select Point 27 from the **Point selection** list.

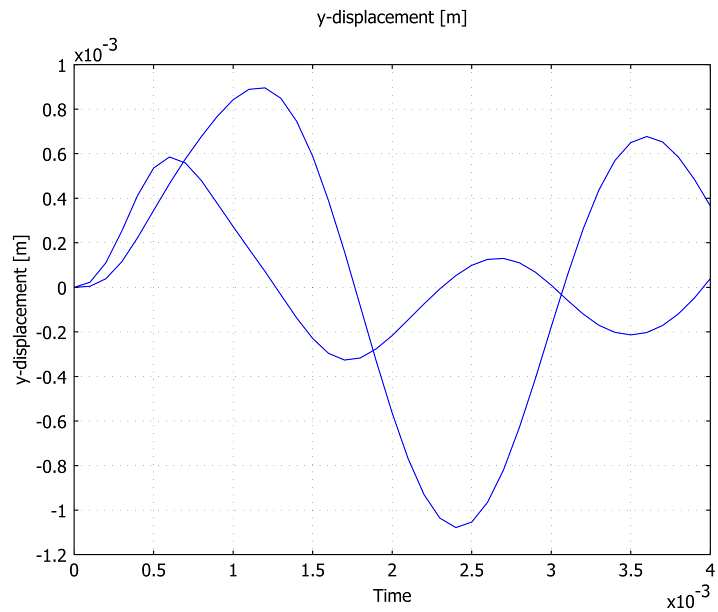
5 Select **x displacement** from the **Predefined quantities** list.

6 Click **Apply** to plot the *x*-displacement.

7 Click the **General** tab and then select the **Keep current plot** check box.

8 Click the **Point** tab and then select **y displacement** from the **Predefined quantities** list.

9 Click **Apply** to plot the y-displacement.



10 Click **OK** to close the dialog box.

Frequency Response Analysis

INTRODUCTION

A frequency response analysis solves for the steady-state response from harmonic excitation loads. The loads can have amplitudes and phase shifts that depend on the excitation frequency, f:

$$F_{freq} = F(f) \cdot \cos\left(2\pi f \cdot t + F_{\mathrm{Ph}}(f) \cdot \frac{\pi}{180}\right)$$

where $F(f)$ is the amplitude and $F_{\mathrm{Ph}}(f)$ is the phase shift of the load.

The purpose of this analysis is to find the transient response from a harmonic load with an excitation frequency in the range 200–600 Hz, which is near the first eigenfrequency found in the eigenfrequency analysis.

Model Library path: `Structural_Mechanics_Module/Tutorial_Models/component_frequency`

MODEL DEFINITION

The geometry, material, and constraints are the same as for the static analysis (see the static description on page 17 for details).

Loads

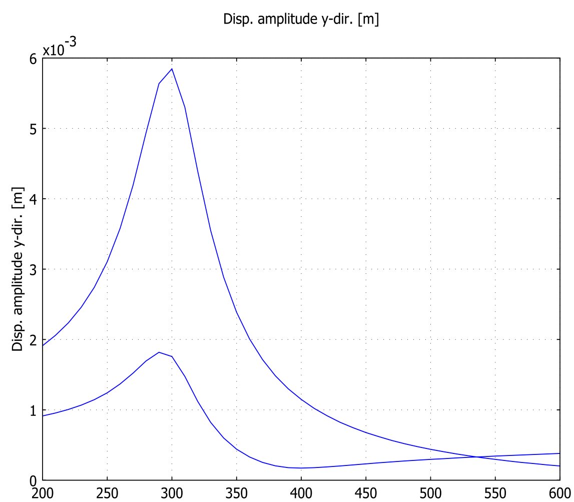
This model uses a harmonic load with an excitation frequency between 200 and 600 Hz on the same edge and with the same amplitude as the static problem.

Damping

Damping is modeled using Rayleigh damping in the same way as for transient analysis (see page 29 for details): $\alpha_{dM} = 189$, $\beta_{dK} = 3.98 \cdot 10^{-5}$

RESULTS

The x- and y-displacements at the bottom end of the edge where the load is applied as a function of excitation frequency appear in the following figure:



There is a peak near the first eigenfrequency where the steady-state response is as much as eight times higher than the static displacement. The amplitude for 500 Hz is below the transient response, which is natural because the damping reduces the transient effects.

MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

The frequency response analysis is described as if it were done after the static analysis described on page 17, so the Plane Stress application mode is already selected.

Geometry Modeling

This was already done in the static analysis (see the static description on page 17 for details).

Physics Settings

Application Mode Parameters

Change the analysis type to frequency response analysis.

1 Choose **Properties** from the **Physics** menu to open the **Application Mode Properties** dialog box.
2 Select **Frequency response** from the **Analysis type** list.

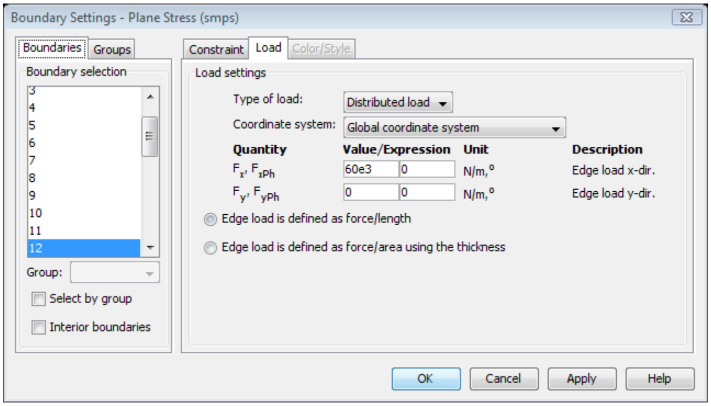
   The **Analysis type** list defines which analysis to perform and which equation to solve.
3 Click **OK**.

Boundary Settings

Use the same amplitude as for the time-dependent analysis without any phase shift, which is the default setting. The left edge is fixed in both directions. In the transient analysis you entered the harmonic excitation load explicitly. The frequency response analysis uses a harmonic assumption and therefore you only need to specify the amplitude of the load, F.

1 Open the **Boundary Settings** dialog box by selecting **Boundary Settings** from the **Physics** menu.
2 Set boundary conditions according to the following table; click **OK** when done.

	BOUNDARY 1		BOUNDARY 12	
Page	Constraint		Load	
	Constraint condition	Fixed	F_x	60e3



Subdomain Settings

The material properties and damping parameters are the same as in the time-dependent analysis on page 22.

Computing the Solution

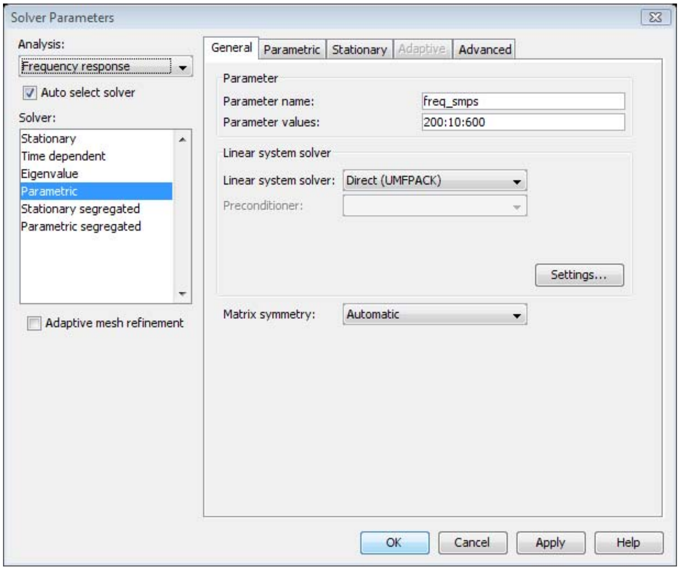
Usually when performing a frequency response analysis you want to sweep over a frequency range. This can be done using the parametric solver. The analysis type controls which solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. This option is enabled as the default, and there is no need to change the solver settings. The parametric solver is the one associated with the frequency response analysis type. Solving for two periods with an excitation frequency of 500 Hz means solving for 0.004 s.

Specify the parametric solver parameters.

1 Select **Solver Parameters** from the **Solve** menu to open the **Solver Parameters** dialog box.

2 Go to the **General** page and enter `freq_smps` in the **Parameter name** edit field.

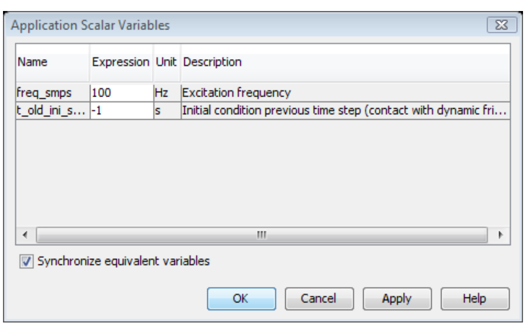
3 Enter `200:10:600` in the **Parameter values** edit field to specify an excitation frequency from 200 Hz to 600 Hz in steps of 10 Hz.



4 Click **OK** to close the **Solver Parameters** dialog box.

If you only want to use the parametric solver to sweep another parameter than the excitation frequency, specify the same **Analysis type** and enter the other parameter in the **Parameter name** edit field. You specify the excitation frequency in the **Application Scalar Variables** dialog box.

5 Select **Scalar Variables** from the **Physics** menu to open the **Application Scalar Variables** dialog box.



You specified a sweep using `freq` as the **Name of the parameter** in the **Solver Parameters** dialog box, which replaces the value of the **Excitation frequency** specified in the **Application Scalar Variables** dialog box.

6 Click **OK** to close the **Application Scalar Variables** dialog box.

7 To compute the solution, either click the **Solve** button (**=**) on the Main toolbar or select **Solve Problem** from the **Solve** menu.

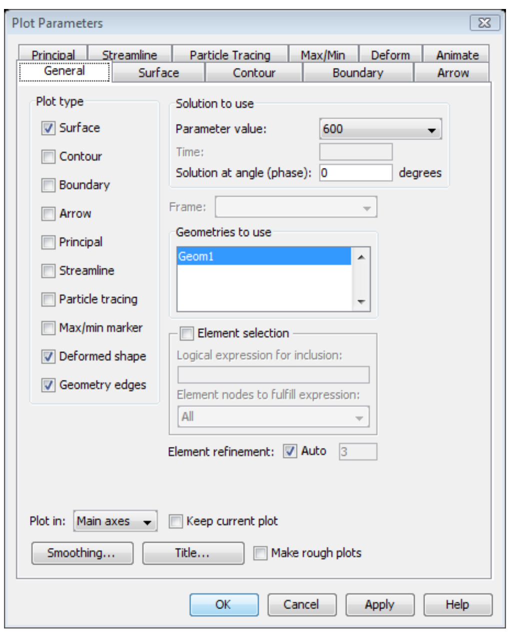
Postprocessing and Visualization

To view the solution at all the different excitation frequencies, open the **Plot Parameters** dialog box, go to the **General** page and select among those frequencies. The default plot shows the von Mises stress for the last excitation frequency in the list.

1 Select **Plot Parameters** from the **Postprocessing** menu.

2 Check **Deformed shape** plot.

3 On the **Surface** page, in the **Surface Data** tab, select **von Mises stress** from the **Predefined quantities** list.



The result of a frequency response analysis is a complex time-dependent displacement field, which can be interpreted as an amplitude, u_{amp}, and a phase angle, u_{phase}. The actual displacement at any point in time is the real part of the solution

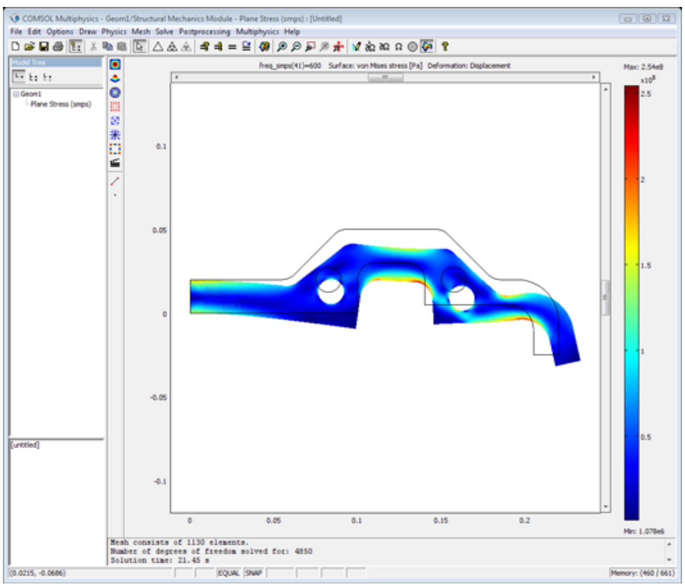
$$u = u_{\text{amp}}\cos(2\pi f \cdot t + u_{\text{phase}})$$

The software can visualize the amplitudes and phases as well as the solution at a specific angle (time). The **Solution at angle** parameter, which you can find on the **General** page, makes this easy. When plotting, COMSOL Multiphysics multiplies the solution by $e^{i\phi}$, where ϕ is the angle in radians that corresponds to the angle specified in degrees in the **Solution at angle** edit field. The plot shows the real part of the evaluated expression

$$u = u_{\text{amp}}\cos(\phi + u_{\text{phase}})$$

The angle ϕ is available as the variable `phase` (radians) and can be used in plot expressions.

4 Click **OK** to close the dialog box.



For a more quantitative view of the frequency evolution of the displacement, plot the x- and y-displacement amplitude at the lower-left corner as a function of the frequency.

1 Select **Domain Plot Parameters** from the **Postprocessing** menu.

2 Go to the **General** page and select all the excitation frequencies in the **Solutions to use** list.

3 Click the **Point** tab (this selects a point plot automatically).

4 Click at the lower-left corner or select Point 27 from the **Point selection** list.

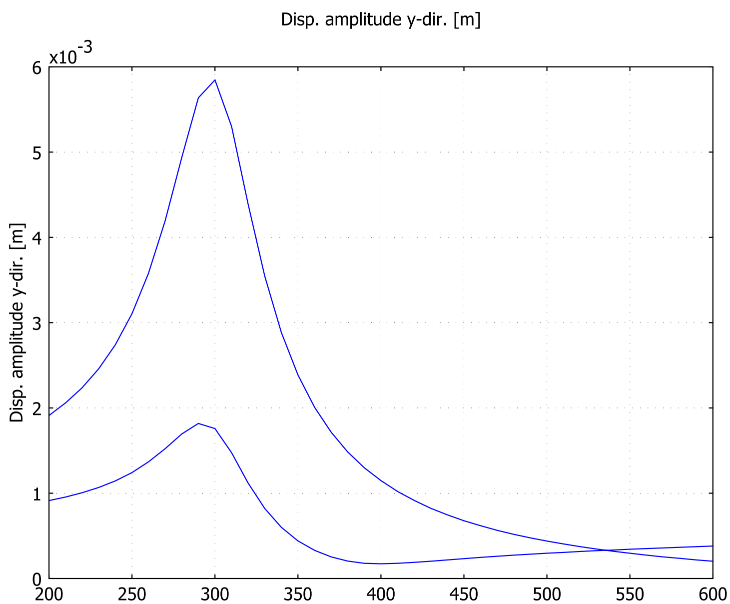
5 Select **Disp. amplitude x-dir.** from the **Predefined quantities** list.

6 Click **Apply** to plot the *x*-displacement amplitude.

7 Click the **General** tab and then select the **Keep current plot** check box.

8 Click the **Point** tab and then select **Disp. amplitude y-dir.** from the **Predefined quantities** list.

9 Click **Apply** to plot the *y*-displacement amplitude.



10 Click **OK**.

Parametric Analysis

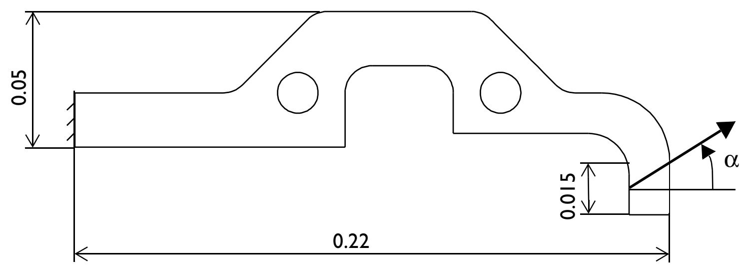
A parametric analysis solves for the static response as a function of a parameter. You freely define the parameter name and what it affects; it can be a material property, a load parameter, or something else.

The purpose of this example is to find the static response as a function of the direction of the force. The force is applied at the same edge as in the static analysis.

Model Library path: `Structural_Mechanics_Module/Tutorial_Models/component_parametric`

MODEL DEFINITION

The geometry, material, and constraints are the same as for the static analysis (see the description on page 17 for details).

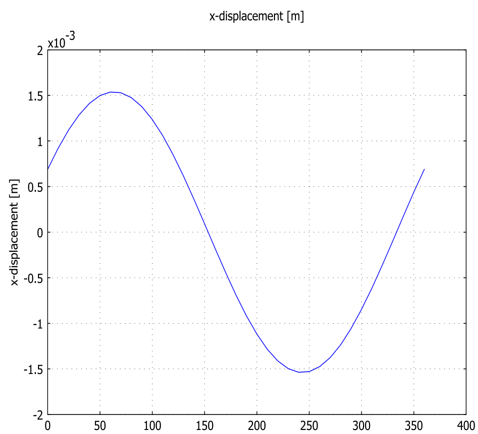


Load

This model uses a static load on the same edge and with the same magnitude as in the static model. The force is free to act in any direction.

RESULTS

The x-displacement at the bottom end of the edge where the load is applied as a function of the direction of the force (α) is shown in the following plot:



MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

The parametric analysis is described as if it were done after the static analysis on page 17, so the Plane Stress application mode is already selected.

Geometry Modeling

This was already done in the static analysis (see the description on page 17 for details).

Physics Settings

Application Mode Parameters

Change the analysis type to parametric analysis.

1 Select **Properties** from the **Physics** menu to open the **Application Mode Properties** dialog box.

   The **Analysis type** list defines which analysis to perform and which equation to solve.

2 Select **Parametric** from the **Analysis type** list; then click **OK**.

Boundary Settings

This model uses the same magnitude as for the static analysis, but the direction α is added as the parameter to vary. The force is divided into its x- and y-components as a function of the force direction, α:

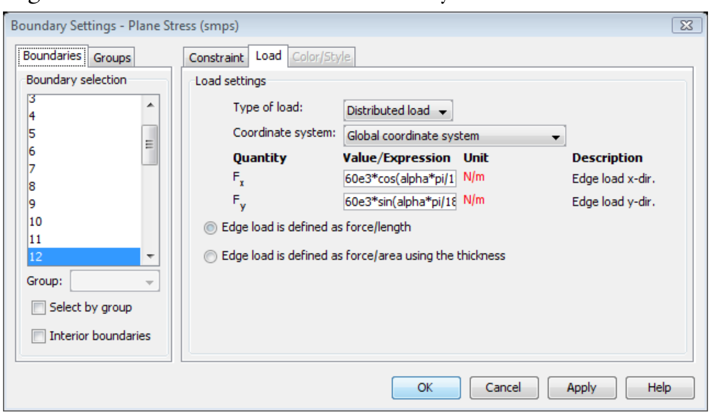
$$F_x = 60 \cdot 10^3 \cos(\alpha)$$
$$F_y = 60 \cdot 10^3 \sin(\alpha)$$

1 Open the **Boundary Settings** dialog box by selecting **Boundary Settings** from the **Physics** menu.

2 Set boundary conditions according to the following table:

	BOUNDARY 1		BOUNDARY 12	
Page	Constraint		Load	
	Constraint condition	Fixed	F_x	`60e3*cos(alpha*pi/180)`
			F_y	`60e3*sin(alpha*pi/180)`

Because the cosine and sine functions take radians as input, a transformation from degrees to radians is done to be able to vary α from 0 to 360°.



Subdomain Settings

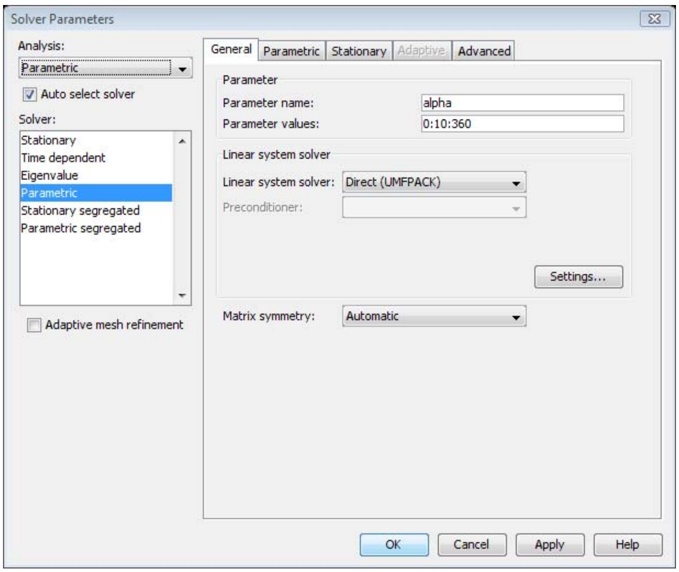
The material properties are the same as in the static analysis on page 22.

Computing the Solution

The analysis type controls which solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. This option is enabled as the default so there is no need to change the solver settings. The parametric solver is the one associated with the parametric analysis type.

Specify the force direction α as the parameter to vary between 0 and 360°.

1 Choose **Solver Parameters** from the **Solve** menu to open the **Solver Parameters** dialog box.
2 In **Parameter**, enter `alpha` in the **Parameter name** edit field.
3 Enter `0:10:360` in the **Parameter values** edit field to specify the direction of the force between 0° to 360° in steps of 10°.



4 Click **OK** to close the **Solver Parameters** dialog box.
5 To compute the solution either click the **Solve** button (**=**) on the Main toolbar or choose **Solve Problem** from the **Solve** menu.

Postprocessing and Visualization

To view the solution for all different directions of the force, open the **Plot Parameters** dialog box, go to the **General** page, then select from the **Parameter value** list. The default plot shows the von Mises stress for the last direction angle `alpha` in the list.

For a more quantitative view of the angle evolution of the displacement, plot the x-displacement at the lower-left corner as a function of the force direction.

1 Select **Domain Plot Parameters** from the **Postprocessing** menu.

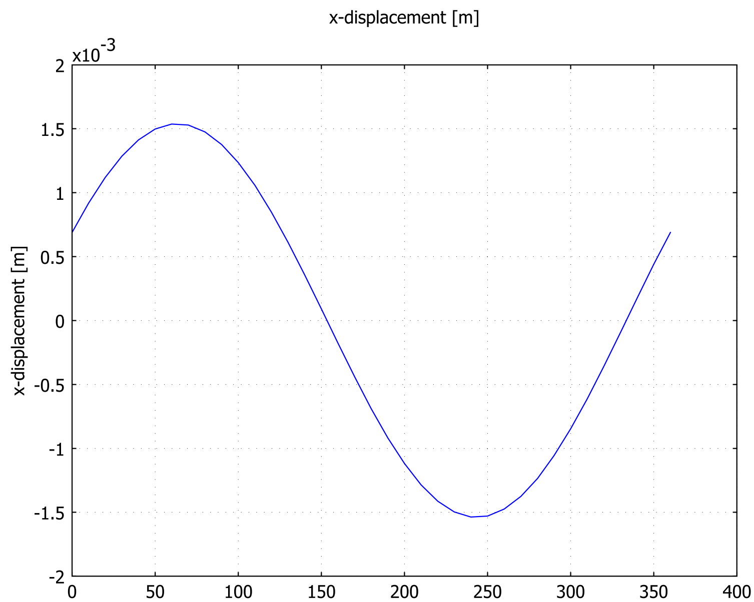
2 Select all load direction angles in the **Solutions to use** list on the **General** page.

3 Click the **Point** tab (this selects a point plot automatically).

4 Click at the lower-left corner or select Point 27 in the **Point selection** list.

5 Select **x-displacement** from the **Predefined quantities** list.

6 Click **Apply** to plot the x-displacement.



7 Click **OK**.

Quasi-Static Transient Analysis

A quasi-static transient analysis solves for the transient response where the dynamics of the structure are static compared to some other much longer time scale. In this example, a transient temperature problem is coupled to the structure, and the temperature problem has a much longer time scale than the dynamics of the structure.

The goal of this analysis is to find out for how long time the component can be exposed to a temperature of 500 °C before the x-displacement of the loaded edge increases by 30% compared to the static displacement without thermal expansion.

Model Library path: `Structural_Mechanics_Module/Tutorial_Models/component_quasi_static`

MODEL DEFINITION

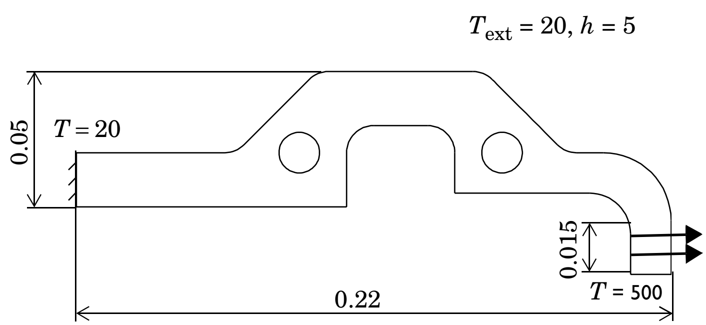
The geometry, material, and constraints are the same as for the static analysis (see the description on page 17 for details).

The boundary conditions for the temperature problem are:

- The left edge has fixed temperature of 20 °C
- The loaded edge has fixed temperature of 500 °C
- All other edges and both sides of the plate are cooled convectively with:
  - External temperature, $T_{inf} = 20$ °C
  - Heat transfer coefficient, $h = 5$ W/(m^2·°C)

The initial conditions is:

- $T = 0$ °C



RESULTS

In the following figure you can see a plot of the x-displacement versus time at the bottom end of the edge where the load is applied.

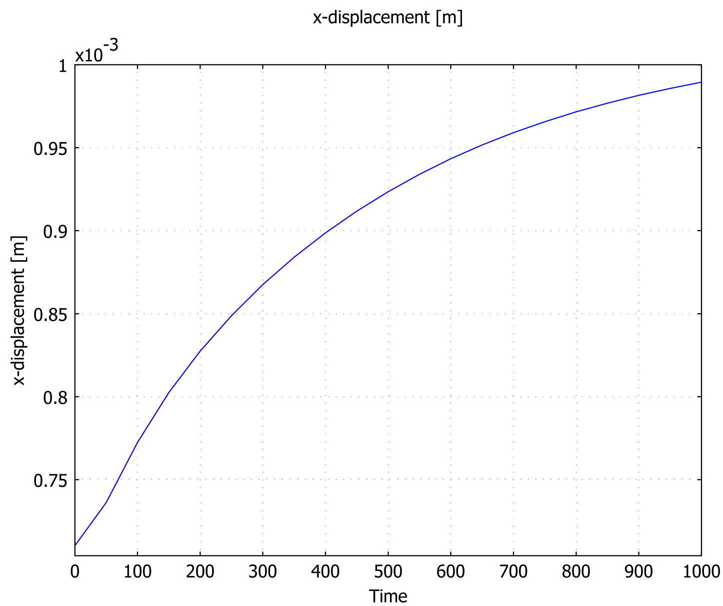


Figure 3-1: The x-displacement at Point 27 as a function of time

From the results you can conclude that it takes 500 s for the displacement to increase by 30%.

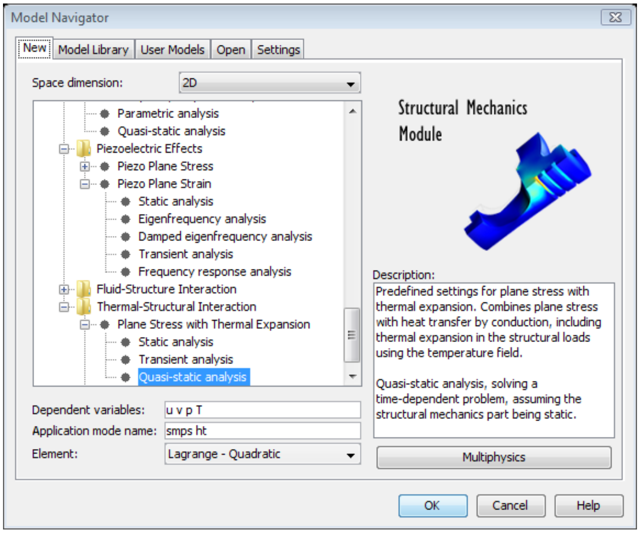
MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

This time the modeling procedure is described as if starting from scratch.

1. Go to the **New** page in the **Model Navigator**, then select **2D** from the **Space dimension** list.

2 Select **Structural Mechanics Module>Thermal-Structural Interaction>Plane Stress with Thermal Expansion>Quasi-static analysis**.



This is a predefined multiphysics coupling that adds both a Plane Stress and a heat transfer application mode (Heat Transfer by Conduction or General Heat Transfer if the license includes the Heat Transfer Module). The predefined multiphysics coupling also sets up the thermal expansion on the **Load** page in the **Subdomain Settings** dialog box for the Plane Stress application mode.

3 Click **OK** to close the **Model Navigator**.

Geometry Modeling

This was described in the static analysis (see the description on page 17 for details).

Physics Settings

Application Mode Properties General Heat Transfer

Skip to the next section if you are using the Heat Transfer by Conduction application mode.

1 Select **Properties** from the **Physics** menu to open the **Application Mode Properties** dialog box.

2 From the **Out-of-plane heat transfer** list box select **Enabled**, then click **OK**.

Boundary Settings Plane Stress

This model uses the same boundary conditions for the Plane Stress application mode as for the static analysis (see page 22 for details).

1 Select **Plane Stress** from the **Multiphysics** menu.

2 Apply boundary settings according to "Boundary Settings" on page 22.

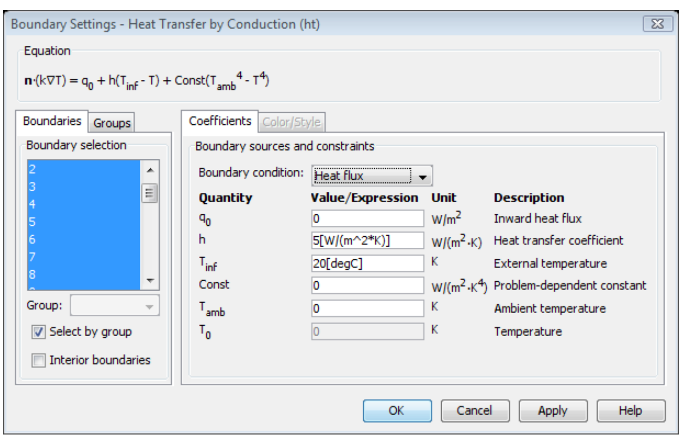
Boundary Settings Heat Transfer

1 Select **Heat Transfer by Conduction** from the **Multiphysics** menu, or **General Heat Transfer** if you are using the Heat Transfer Module.

2 Set boundary conditions for the heat transfer application mode according to the following table. Click **OK** when done.

An easy way to apply the settings is if you first select all boundaries, then apply the **Heat flux** boundary condition according to the last two columns of the table. Finish by changing the settings for boundary 1 and 12 according to the table.

	BOUNDARY 1		BOUNDARY 12		BOUNDARIES 2–11, 13–30	
Boundary condition	Temperature		Temperature		Heat flux	
	T_0	20[degC]	T_0	500[degC]	T_{inf}	20[degC]
					h	5[W/(m^2*K)]



Subdomain Settings Plane Stress

The material properties for the Plane Stress application mode are the same as in the static analysis on page 22.

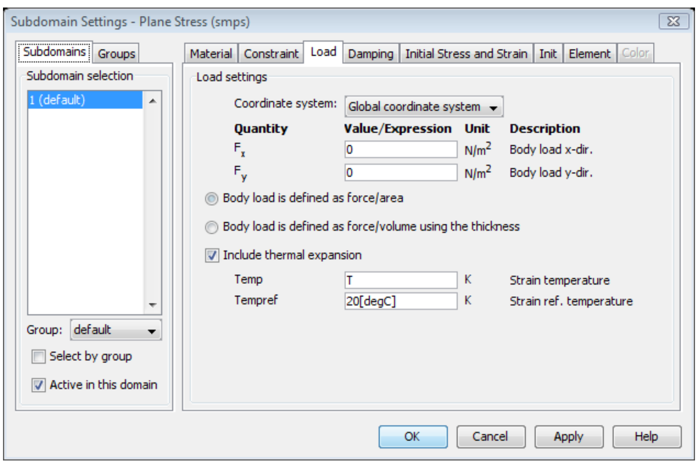
Specify the reference temperature in the Plane Stress application mode. The temperature coupling is already defined because you used the predefined multiphysics coupling node in the Model Navigator.

1 Select **Plane Stress** from the **Multiphysics** menu.

2 Set the reference temperature for the Plane Stress application mode according to the following table:

SUBDOMAIN	1	
Page	Load	
	`Tempref`	`20[degC]`

You can see that the variable `T`, which is the temperature from the heat transfer application mode, is entered by default in the **Temp** edit field.



Subdomain Settings Heat Transfer by Conduction

Skip to the next section if you are using the General Heat Transfer application mode from the Heat Transfer Module.

Specify the material properties for the heat transfer application mode. The cooling of the surfaces is the same as for the boundaries, but you must transform it into cooling per volume, taking into account the thickness of the plate and that it has two sides.

Instead of 5, h_{trans} is 2500 coming from division by the thickness and multiplication by 2.

1 Select **Heat Transfer by Conduction** from the **Multiphysics** menu.

2 Set the subdomain settings for the heat transfer application mode according to the following table. Click **OK** when finished.

	SUBDOMAIN 1	
Page	Physics	
	Library material	Structural steel
	T_{ext}	20[degC]
	h_{trans}	2500[W/(m^3*K)]

Subdomain Settings General Heat Transfer

Skip to the next section if you are using the Heat Transfer by Conduction application mode.

1 Select **General Heat Transfer** from the **Multiphysics** menu.

2 Set the subdomain settings according to the following table, then click **OK**.

	SUBDOMAIN 1	
Page	Conduction	
	Library material	Structural steel
Page	Out-of-Plane	
	h_u	5[W/(m^2*K)]
	h_d	5[W/(m^2*K)]
	$T_{ext,u}$	20[degC]
	$T_{ext,d}$	20[degC]
	d_z	4[mm]

Computing the Solution

The analysis type controls which solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. The option is enabled as the default so there is no need to change the solver settings. The time-dependent solver is the one associated with the quasi-static transient analysis type.

Specify the time-dependent solver parameters:

1 Select **Solver Parameters** from the **Solve** menu to open the **Solver Parameters** dialog box.

2 Enter `0:50:1000` in the **Times** edit field. This means that the software saves the solution every 50 s during the total solution time of 1000 s. This does not influence the time-stepping length, only the times when the solution is saved.

3 Click **OK**.

4 To compute the solution either click the **Solve** button (**=**) on the Main toolbar or select **Solve Problem** from the **Solve** menu.

Postprocessing and Visualization

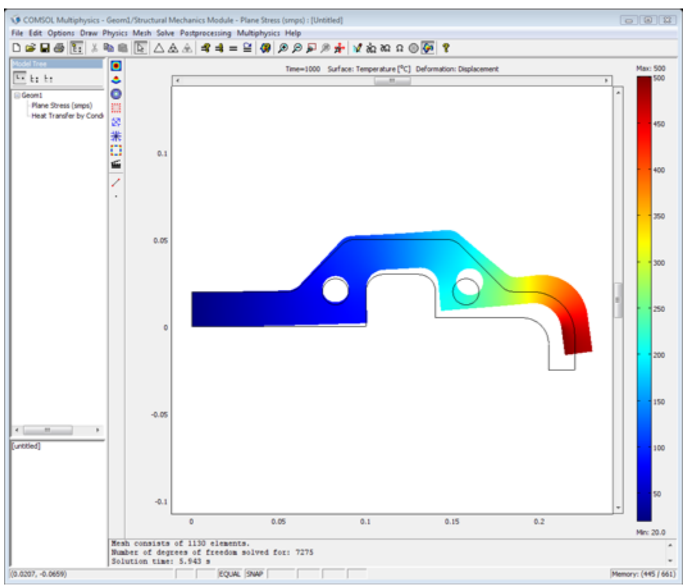
Look at the final temperature distribution together with the deformed geometry.

1 Select **Plot Parameters** from the **Postprocessing** menu.

2 Select the **Deformed shape** check box in the **Plot type** area.

3 Go to the **Surface** page and select **Heat Transfer by Conduction (ht)>Temperature** (or **General Heat Transfer (htgh)>Temperature**) from the **Predefined quantities** list.

4 Click **OK** to close the **Plot Parameters** dialog box.



For a quantitative view of the time evolution of the displacement, plot the x-displacement at the lower-left corner as a function of time.

1 Select **Domain Plot Parameters** from the **Postprocessing** menu.

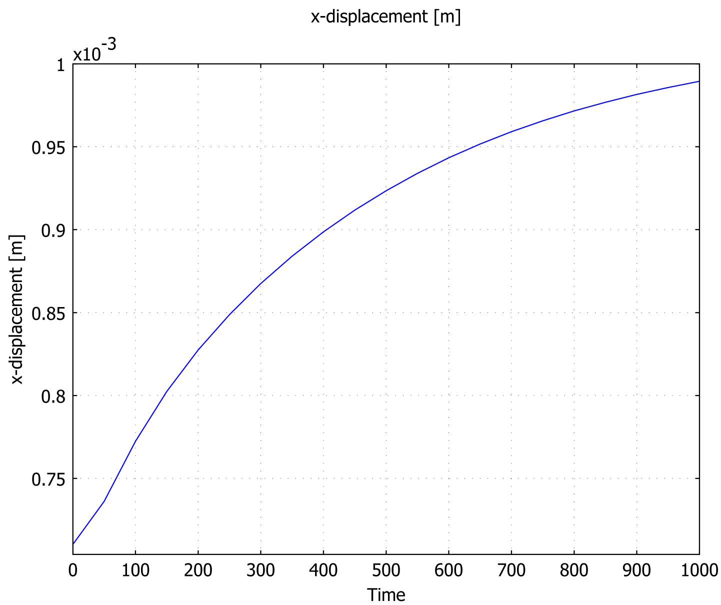
2 Go to the **General** page and select all the time steps in the **Solutions to use** list.

3 Click the **Point** tab (this selects a point plot automatically).

4 Click at the lower-left corner or select Point 27 from the **Point selection** list.

5 Select **x-displacement** from the **Predefined quantities** list.

6 Click **Apply** to plot the x-displacement.



7 Click **OK** to close the **Domain Plot Parameters** dialog box.

4

Structural Mechanics Modeling

The objective of this chapter is to give you an insight on how to approach the modeling of various structural mechanics problems. The contents cover subjects like loads and constraints, units, reaction forces, and material models.

Loads

An important aspect of structural analysis is the formulation of the forces applied to the modeled structure. You have the freedom of using custom expressions, predefined or user-defined coordinate systems, and even variables from other application modes.

You can apply loads on the **Load** page of either one of the following dialog boxes: **Subdomain Settings**, **Boundary Settings**, **Edge Settings**, or **Point Settings**. You access these from the **Physics** menu or the **Model Tree**. This *User's Guide* includes a detailed description of the above functionality for each application mode in the Structural Mechanics Module. Use the Table 4-1 below to locate the relevant page.

TABLE 4-1: LOAD SETTINGS FOR APPLICATION MODES IN THE STRUCTURAL MECHANICS MODULE

APPLICATION MODE	LOAD SETTINGS
Continuum Application Modes	page 205
Mindlin Plates	page 247
Shells	page 312
Beams	page 270
Trusses	page 293
Piezoelectric application modes	page 344

Units, Orientation, and Visualization

USING UNITS

You can enter loads in any unit, independently of the base unit system in the model, because COMSOL Multiphysics automatically converts any unit to the base unit system. To use the feature for automatic unit conversion, enter the unit in square brackets, for example, `100[lbf/in^2]`. You can read more about unit systems in "Using Units" on page 183 of the *COMSOL Multiphysics User's Guide*.

PREDEFINED AND CUSTOM COORDINATE SYSTEMS

In the Structural Mechanics Module, two predefined coordinate systems are always available when you specify loads. These are the global coordinate system and the local tangent and normal coordinate system to the boundary.

Custom coordinate systems are also available and are useful, for example, to specify a load in any direction without breaking it up into components. To set up a coordinate system open the **Coordinate Systems** dialog box from the **Options** menu. For a detailed explanation of coordinate systems, see Chapter 6, "Coordinate Systems," on page 144.

VISUALIZING LOADS

A convenient check for load orientation is to activate the display of load symbols on the geometry. You can do this on the **Visualization** page of the **Preferences** dialog box, which you access from the **Options** menu. Load symbols are listed on page 154. You can also read about visualization settings on page 119 of the *COMSOL Multiphysics User's Guide*.

Another way to visualize loads is to create plots of the global force or surface tractions during postprocessing.

Load Cases

Similar to the familiar concept of load cases, but more powerful, is the parametric solver available in all the application modes. You can select the parametric solver from the **Solver Parameters** dialog box which you access from the **Solve** menu. Here you can either select **Parametric** from the **Analysis** list or just select **Parametric** from the **Solver** list, which works together with all of the analysis types. On the **General** page, you then name your parameter and define a list of values. The parameter defined here is available in any expression. You can easily control the magnitude, distribution, and even location of loads.

A good example on how to set up expressions for controlling position and distribution of loads, with the help of the parametric solver, is the model in the *Structural Mechanics Module Model Library.*

Singular Loads

In reality, loads always act on a finite area. However, in a model you can define a load on a point or an edge, which leads to singularities. The reason for this is that points and lines have no area, so the stress becomes infinite. Because of the stress singularity, there are high stress values in the area surrounding the applied load. The size of this area and the magnitude of the stresses depend on both the mesh and the material properties. The stress distribution at locations far from these singularities is unaffected according a to a well-known principle in solid mechanics, the St. Venant's principle. It states that for an elastic body, statically equivalent systems of forces produce the same stresses in the body, except in the immediate region where the loads are applied.

The Figure 4-1 on page 66 shows a plate with a hole in plane stress loaded with a distributed load and a point load of the same magnitude. The mesh consists of triangular elements with quadratic shape functions. The high stress around the point load is dissipated within the length of a few elements for both mesh cases. The stresses in the middle of the plate and around the hole are in agreement for the distributed load and the point load. The problem is that due to the high stress around the singular load it is easy to overlook the high stress region around the hole. When you apply the point load, you have to manually set the range for the stress plot to get the same visual feedback of the high stress region around the hole in the two cases. This is because the default plot settings automatically set the range based on the extreme values of the expression that is plotted.

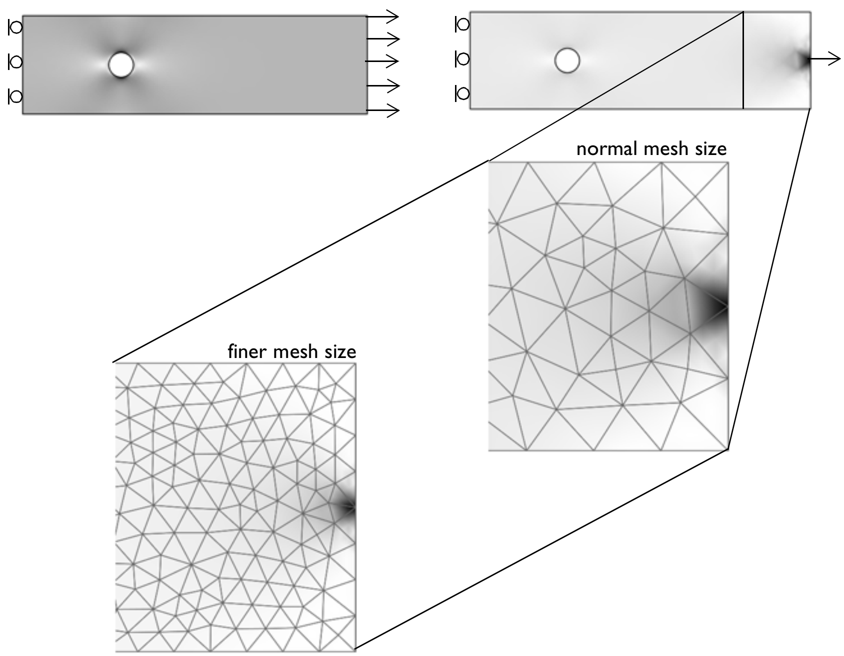


Figure 4-1: A plate with a hole subject to a distributed load (left) and a point load (right).

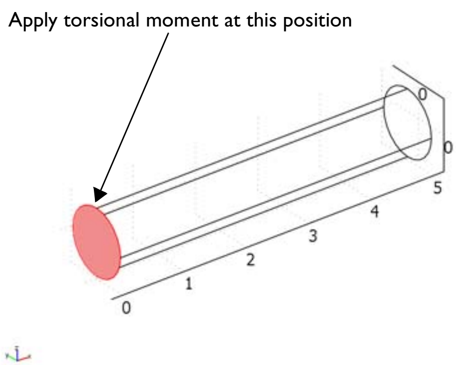
Despite these findings it is good modeling practice to avoid singular loads because it is difficult to estimate the size of the singular region. In the Structural Mechanics Module it is possible to define loads on all boundary types. However, avoid singular loads altogether with nonlinear and elasto-plastic material models.

Moments in the Continuum Application Modes

The continuum application modes do not have rotational degrees of freedom, which makes the specification of moments somewhat cumbersome. To specify moments, you can apply a stress distribution which corresponds to the moment.

EXAMPLE: TORSIONAL MOMENT ON A CYLINDER

The following steps show how to apply a torsional moment at one end of a cylindrical axle in the following figure.



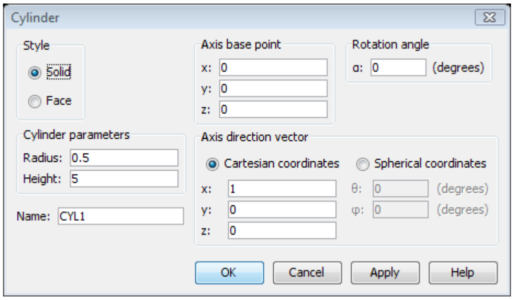
1 In the **Model Navigator** select **3D** from the **Space dimension** list.

2 In the list of application modes select **Structural Mechanics Module>Solid, Stress-Strain>Static analysis**; then click **OK**.

Continue by creating a cylinder:

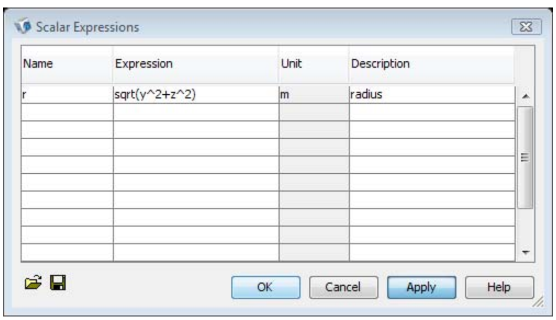
3 In the **Draw** toolbar click the **Cylinder** button.

4 In the dialog box that opens, enter `0.5` as the **Radius**, `5` as the **Height**, and define the axis of the cylinder along the *x*-axis by entering `1` in the **x** edit field and `0` in the **y** and **z** edit fields in the **Axis direction vector** area. Click **OK**.



Next define the radial location as a scalar expression, which you can later use in the load expression.

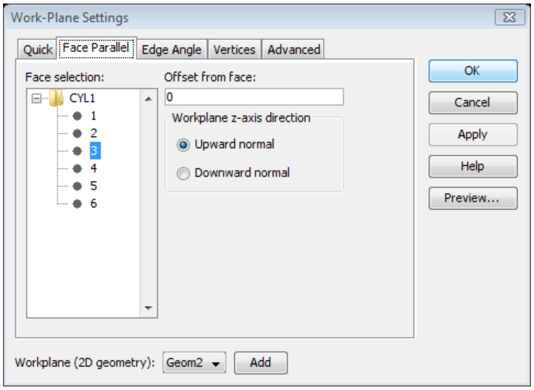
5 Choose **Options>Expressions>Scalar Expressions**. Enter `r` in the **Name** column and `sqrt(y^2+z^2)` in the **Expression** column. Click **OK**.



Create a work plane that you can use to define a cylindrical coordinate system.

6 Select **Draw>Work-Plane Settings**.

7 On the **Face Parallel** page in the **Work-Plane Settings** dialog box select Face 3 from the **Face selection** list. Click **OK**.



8 A new geometry appears with the work plane that you just created. Switch back to the geometry containing the cylinder by clicking the Geom1 page in the Drawing area.

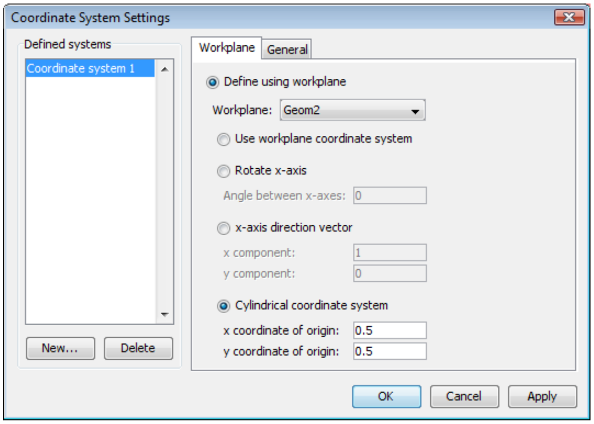
Continue by defining a cylindrical coordinate system with the help of the work plane you just created.

9 Select **Options>Coordinate Systems**. Click **New** in the **Coordinate System Settings** dialog box.

10 Click **OK** in the **New Coordinate System** dialog box.

11 Back in the **Coordinate System Settings** dialog box make sure that **Define using work plane** is selected as well as **Geom2** in the **Work plane** list.

12 Click the **Cylindrical coordinate system** button. Enter `0.5` for both the x- and y-coordinates of the origin, because these are the coordinates of the center of the circular face in the coordinate system of the work plane. Click **OK**.



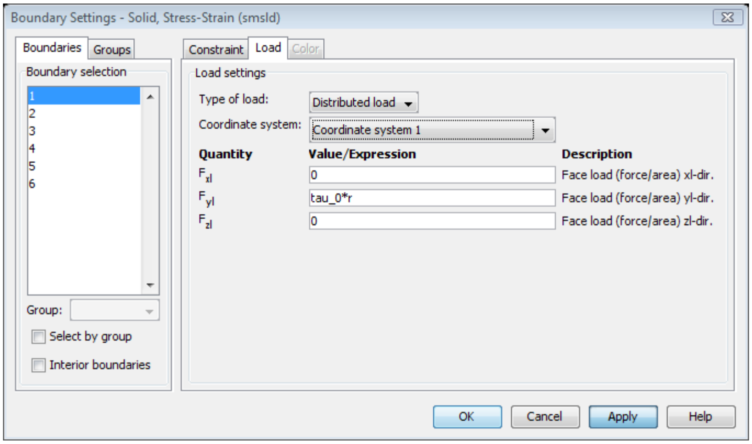
Now you can define a shear stress distribution in the tangential direction, which is zero at the center and reaches its maximum at the surface of the axle.

13 Select **Physics>Boundary Settings** to open the **Boundary Settings** dialog box.

14 Select Boundary 1 from the **Boundary selection** list.

15 On the **Load** page select **Coordinate system 1** (the one you just created) from the **Coordinate system** list.

16 Enter `tau_0*r` in the $\mathbf{F_{yl}}$ edit field. Click **OK**.

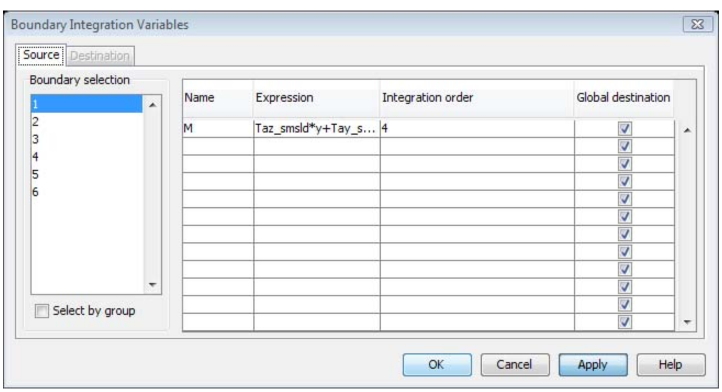


Because the integral of the stress distribution over the boundary must equal the moment to satisfy equilibrium, you can easily specify a value for the moment based on the stress distribution.

17 Define the boundary integral for the moment. Select **Options>Integration Coupling Variables>Boundary Variables**.

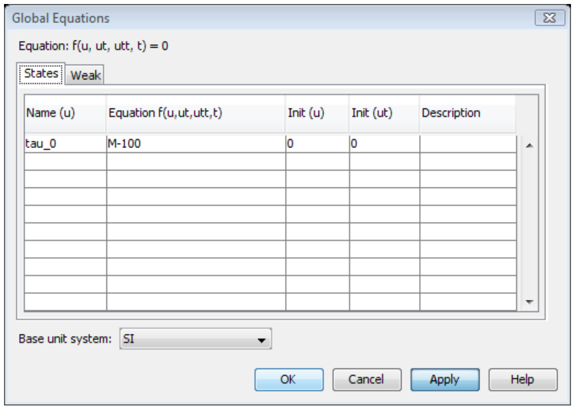
18 In the dialog box that opens select Boundary 1, which is the one where you have defined the load.

19 Enter `M` in the **Name** column and `Taz_smsld*y+Tay_smsld*z` in the **Expression** column. Click **OK**.



20 In this last step you specify a value for the moment through defining an equilibrium equation. Choose **Physics>Global Equations** to open the **Global Equations** dialog box.

21 Enter `tau_0` in the **Name** column and `M-100` in the **Equation** column. The value 100 is the magnitude of the applied moment. This entry defined the equation $M - 100 = 0$, that is, $M = 100$.



22 Constrain the other end face of the cylinder by fixing it before solving the model.

After solving the model you can visualize the moment on the cylinder by an arrow plot of the global force on the boundaries.

23 Click the **Plot Parameters** button on the Main toolbar.

24 On the **General** page clear the **Slice** check box and select the **Arrow** check box.

25 Click the **Arrow** tab to switch to the **Arrow** page.

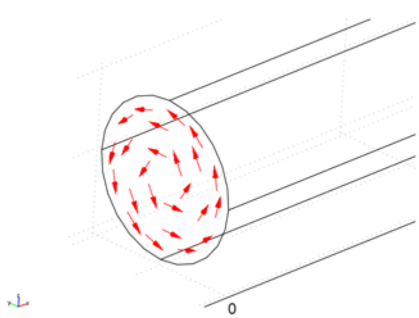
26 From the **Plot arrows on** list box select **Boundaries**.

27 From the **Predefined quantities** list box select **Global force**.

28 From the **Arrow type** list box select **3D arrow**.

29 From the **Arrow length** list box select **Normalized**.

30 Click **OK**.



Follower Loads

Follower loads are defined with respect to the geometry and, as the geometry deforms locally, the orientation of the load changes. The use of follower loads is meaningful only if you have large deformations in your model and have activated the Large Deformation option in the **Application Mode Properties** dialog box. To define a load as a follower load you can select **Follower load** in the **Load type** list box on the **Load** page of the **Boundary Settings** dialog box.

Acceleration Loads

Acceleration loads can be found, for example, in the structural mechanics analysis of an airplane seat. Acceleration or deceleration of the aircraft produces a force that an accurate simulation must include. Because you can use expressions when specifying loads, it is easy to model acceleration loads. In the case of the airplane seat, you define the acceleration, `acc_x`, in the **Constants** dialog box. Then for the appropriate subdomains simply enter `rho_smsld*acc_x` in the **Body load x dir.** edit field on the **Load** page of the **Subdomain settings** dialog box. The density `rho_smsld` is already defined for the material and refers to the corresponding edit field. In a similar manner it is also possible to specify gravity loads.

For modeling rotating parts under static conditions, you can use centrifugal acceleration loads. The body load in the radial direction is

$$K_r = \rho\omega^2 r, \tag{4-1}$$

where ρ is the density of the material, ω is the angular frequency, and r is the radius.

EXAMPLE: A ROTATING DISK

This example describes how to specify a body load according to Equation 4-1, using a cylindrical coordinate system. The model is that of a disk welded on a shaft, which rotates with a constant angular velocity of 90 rad/s.

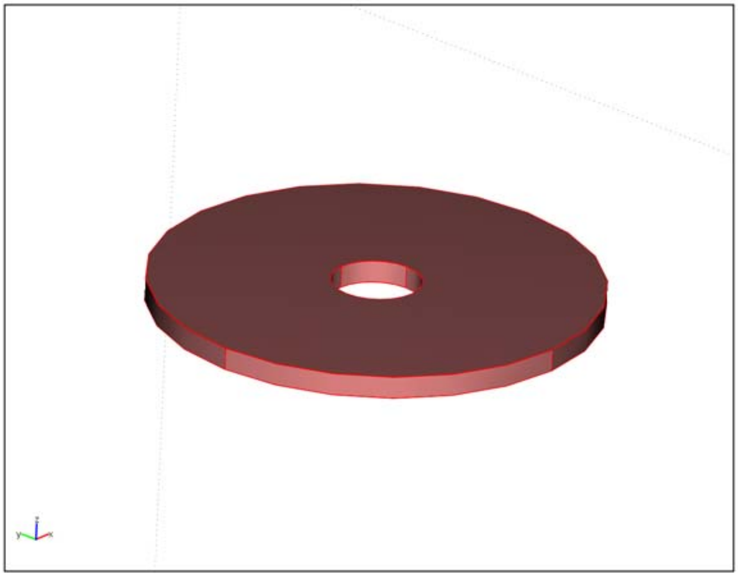
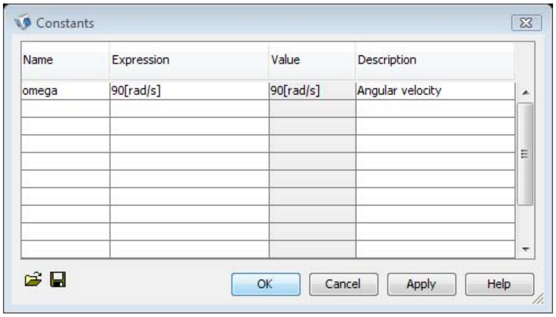


Figure 4-2: The model geometry.

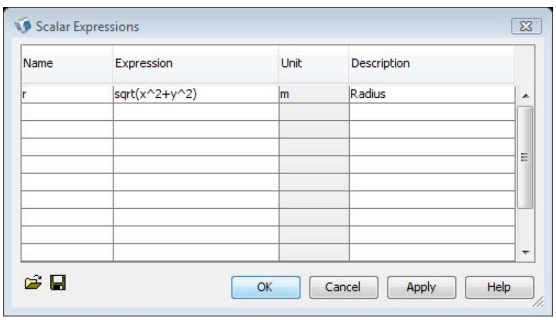
1. In the **Model Navigator**, select **3D** from the **Space dimension** list.
2. Select **Structural Mechanics Module> Solid, Stress-Strain**, then click **OK.**
3. Create the geometry by first drawing two cylinders, then taking the difference between them.
4. Click the **Cylinder** button on the Draw toolbar.
5. Enter `0.1` in the **Radius** edit field, and `0.01` in the **Height** edit field. Click **OK**.
6. Create one more cylinder with a radius of `0.02` and a height of `0.01`.
7. Select both objects by typing Ctrl+A on the keyboard.
8. Click the **Difference** button on the Draw toolbar to create the final geometry.
9. Select **Constants** from the **Options** menu.

10 Enter `omega` in the **Name** column and `90[rad/s]` in the **Expression** column.When finished click **OK**.



11 Select **Options>Expressions>Scalar Expressions**.

12 Enter `r` in the **Name** column, and `sqrt(x^2+y^2)` in the **Expression** column. Click **OK**.



Next set up the cylindrical coordinate system by first defining a work plane.

13 Select **Draw>Work-Plane Settings**.

14 Select the **x-y** radio button on the **Quick** page and click **OK**.

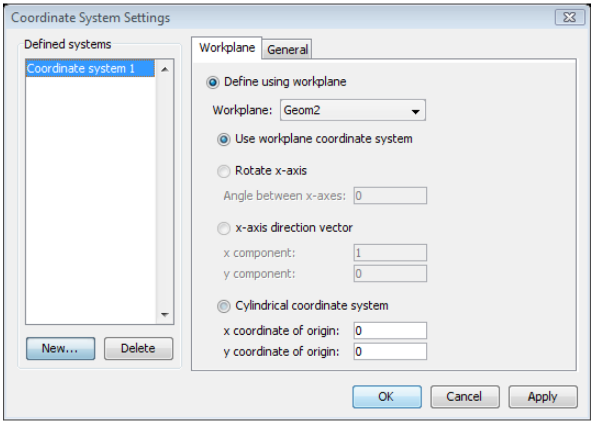
15 The new workplane is now the active geometry. Switch back to your initial geometry, by clicking the **Geom1** tab at the top of the drawing area.

16 Select **Options>Coordinate Systems** to open the **Coordinate System Settings** dialog box.

17 Click **New**, then click **OK** to accept the default name.

18 On the **Workplane** page, click the **Define using workplane** button. From the **Workplane** list, select **Geom2**.

19 Select the **Cylindrical coordinate system** and click **OK**.



You can now define the radial body load, by using the defined coordinate system.

20 Open the **Subdomain Settings** dialog box from the **Physics** menu.

21 Select the only subdomain and then go to the **Load** page.

22 From the **Coordinate system** list, select the coordinate system you have just defined.

23 Enter `rho_smsld*omega^2*r` in the $\mathbf{F_{xl}}$ edit field, and then click on **OK**.

Before solving the model you need to constrain the disk.

24 From the **Physics** menu, select **Boundary Settings**.

25 From the list of boundaries, select boundary 5, 6, 8, and 9.

26 Select **Prescribed displacement** from the **Constraint condition** list, and then select **Tangent and normal coord. sys.** $\mathbf{(t_1,t_2,n)}$, from the **Coordinate system** list.

27 Select the $\mathbf{R_n}$ check box, and then click **OK**.

28 From the **Physics** menu select **Point settings**.

29 Select point 3, then select the $\mathbf{R_z}$ check box.

30 Click **OK**.

31 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.

32 From the list of predefined mesh sizes, select **Finer**, and then click **OK.**

33 Click the **Solve** button on the Main toolbar to compute the solution.

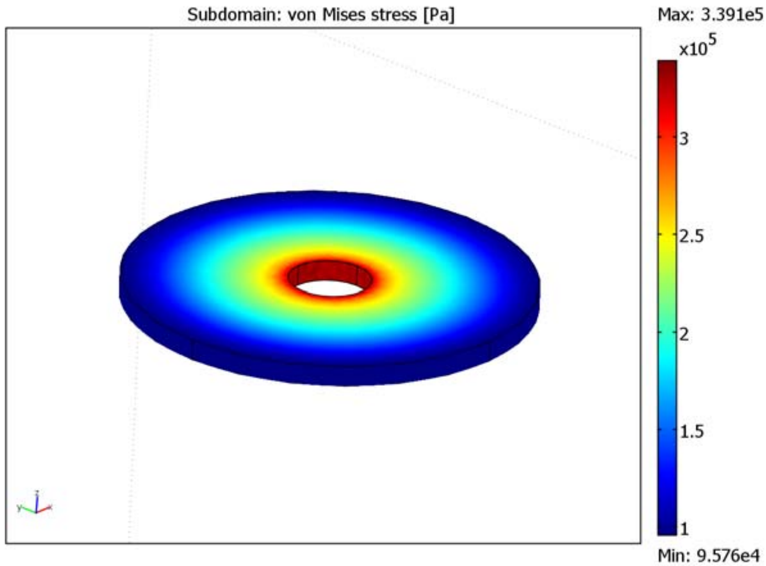


Figure 4-3: The von Mises stress distribution.

34 Click the **Plot Parameters** toolbar button.

35 In the **Plot type** area, clear the **Slice** check box and select the **Subdomain** check box.

36 Click **OK** to create the plot shown in Figure 4-3.

Temperature Loads—Thermal Expansion

When performing thermal expansion analysis, you specify temperature loads by entering a temperature and a reference temperature on the **Load** page of the **Subdomain Settings** dialog box. You can enter a constant temperature as well as an analytic expression that can depend on the coordinates or dependent variables. More details are available in the descriptions for each application mode (see Table 4-1 on page 64).

When you use a separate application mode to model heat transfer in the material, the entry for the temperature is the dependent variable for the temperature from that application mode, typically T. Read more about how to couple heat transfer analysis with structural mechanics analysis on page 114.

The Structural Mechanics Module also includes a predefined multiphysics coupling to a heat transfer application mode. See "Thermal-Structure Interaction" on page 354 for more information.

Total Loads

You can specify a load either as a distributed load per unit length, area, or volume, or as a total force to be uniformly distributed on a boundary.

When you apply a distributed load on a boundary, the Structural Mechanics Module provides a way to check that the total force is correct without having to solve the model. After you have entered the load, choose **Solve>Get Initial Value**. Then choose **Postprocessing>Boundary Integration**. In the dialog box that appears, you can select a boundary and from the **Predefined quantities** list select a face load in 3D or an edge load in 2D. Click **Apply** to display the value of the integral, which is the value of the total force in the selected direction, in the message log.

Similarly you can check the total body load by subdomain integration of the appropriate component of a body load.

Constraints

Defining the proper constraints for structural mechanics models is just as important as defining the loads. Together they make up the boundary conditions of a model. The Structural Mechanics Module provides many useful features to define various types of constraints. Besides all the predefined options, you can use any expressions to define constraints of your choice.

From the **Physics** menu, you can access the dialog boxes where you can define constraints. Depending on your model these can be one or more of the **Subdomain Settings**, **Boundary Settings**, **Edge Settings** or **Point Settings** dialog boxes. Each application mode description includes a complete list of available options for constraint settings, use the following table to find the appropriate pages.

APPLICATION MODE	CONSTRAINT SETTINGS
Continuum Application Modes	page 201
Mindlin Plates	page 244
Shells	page 307
Beams	page 265
Trusses	page 289
Piezoelectric application modes	page 342

Orientation and Visualization

You can specify constraints in both global and local coordinate systems. Beside these, you can use any coordinate system that you have previously defined in the **Coordinate System Settings** dialog box, which you access from the **Options** menu. Coordinate systems are further explained on page 144.

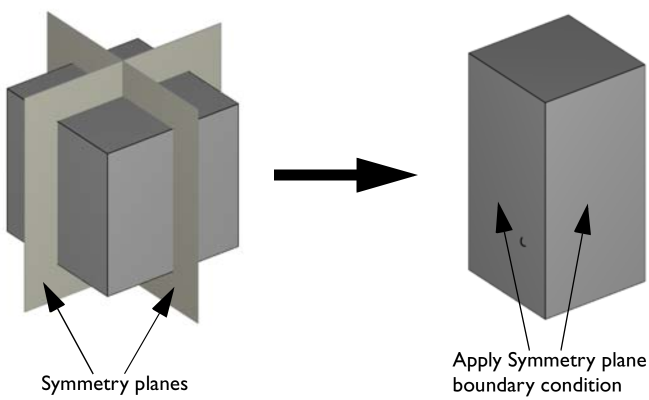
When you turn on the visualization of load and constraint symbols from the **Preferences** dialog box, COMSOL Multiphysics displays all applied constraints with symbols on the geometry. See page 156 for a list of the symbols for constraints.

Symmetry Constraints

In many cases you can use symmetry of the geometry and loads to your advantage in modeling. Symmetries can often greatly reduce the size of a model and hence reduce the memory requirements. When a structure exhibits axial symmetry, you can use the axisymmetric application modes. A solid that you can generate by rotating a planar shape about an axis is said to have axial symmetry.

For other types of symmetry, you can use many of the predefined symmetry constraints available in the user interface of the Structural Mechanics Module. This means that you do not have to enter any expressions—instead just select the type of constraint you want to apply from a list.

If the geometry exhibits two symmetry planes, as shown in the figure below, you can model a quarter of the geometry by selecting **Symmetry plane** from the **Constraint condition** list on the **Constraint** page in the **Boundary Settings** dialog box for the two selected surfaces.



As mentioned earlier not only the symmetry of the geometry but also that of the load is important in selecting the correct constraints for your model. Figure 4-4 on page 81 illustrates symmetric and antisymmetric loading of a symmetric geometry. When modeling half of the geometry, the correct constraint for the face at the middle of the object would be **Antisymmetry plane** in the case of antisymmetric loading and **Symmetry plane** in the case of symmetric loading of the object.

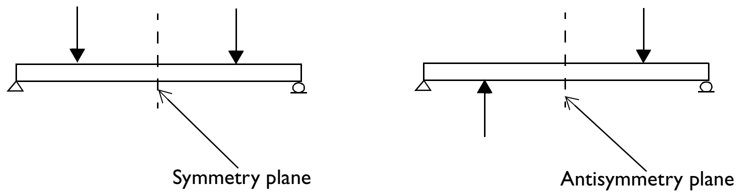


Figure 4-4: Symmetry plane (left) and antisymmetry plane (right).

You can often take advantage of symmetry by using the superposition principle and thus reduce the size of models. The superposition principle states that for linear elastic materials you can solve separately for different load cases and superpose or add the solutions afterwards. For symmetric objects, it is possible to separate any load into a symmetric load and antisymmetric load. Thus, you can solve two models on half of the geometry and later superpose or add the solutions. Read about how to apply this technique to analyze a wheel rim of a car in the description of the model "Automobile Wheel Rim" on page 108 of the *Structural Mechanics Model Library*.

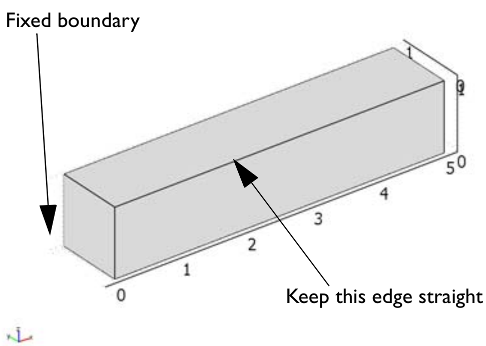
Kinematic Constraints

Kinematic constraints are equations that control the motion of solids, faces, edges, or points. Select **Prescribed displacement** from the **Constraint Condition** list to enter expressions for constraints. You can define the equations using both predefined coordinate systems and custom coordinate systems. Special constraints, for instance to keep an edge of body straight or to make a boundary rotate, require such constraint equations.

EXAMPLE: STRAIGHT EDGE CONSTRAINT BY EQUATIONS

The following short example shows how to implement constraint equations to keep an edge of a solid body straight during deformation. The three linear constraints

(Equation 10-18) are derived starting on page 281 and are used to keep truss elements straight in the 3D truss application modes.

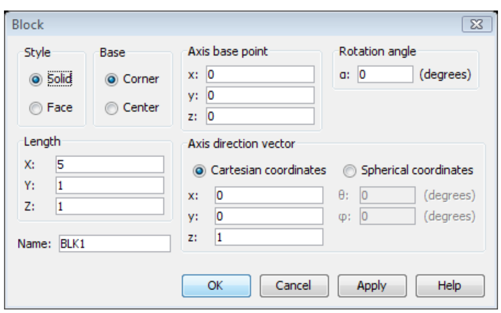


1 In the **Model Navigator** select **3D** from the **Space dimension** list.

2 In the list of application modes select **Structural Mechanics Module>Solid, Stress-Strain>Eigenfrequency analysis**; then click **OK**.

Continue by creating a bar with a rectangular cross section.

3 On the **Draw** toolbar click the **Block** button. In the dialog box that appears enter 5 in the **X** edit field under the **Length** label. Click **OK**.



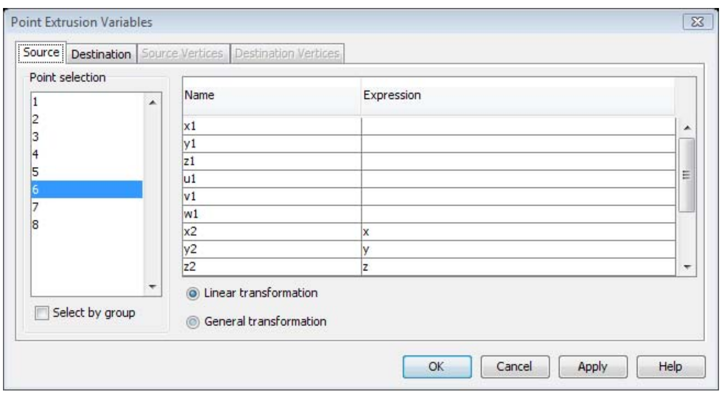
Continue by defining point extrusion variables, which you can use in the constraint equation.

4 Select **Options>Extrusion Coupling Variables>Point Variables**, which opens the **Point Extrusion Variables** dialog box.

5 From the **Point selection** list select Point 2. Enter the expressions belonging to this point from the following table in the **Name** and **Expression** columns. Make sure to select **General transformation** for each variable.

POINT 2		POINT 6	
NAME	**EXPRESSION**	**NAME**	**EXPRESSION**
x1	x	x2	x
y1	y	y2	y
z1	z	z2	z
u1	u	u2	u
v1	v	v2	v
w1	w	w2	w

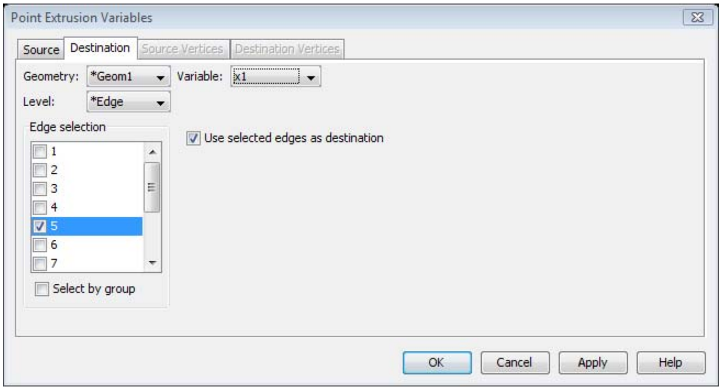
6 Select Point 6 and enter the variables and expressions listed for this point. Select **General transformation** for each of these variables as well.



Next you define the destination for the variables. This is the edge to which you apply the constraint equation.

7 On the **Destination** page select **x1** in the **Variable** list, then select **Edge** in the **Level** list. Now select the check box in front of Edge 5 in the **Edge selection** list.

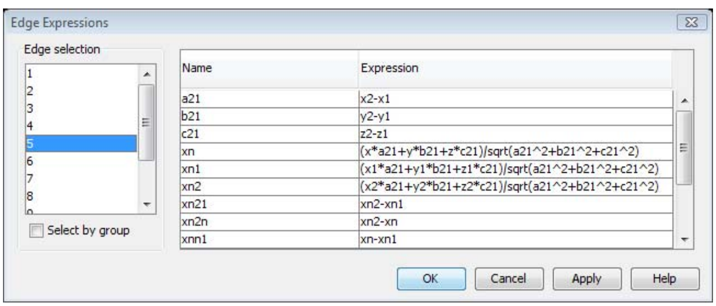
8 Repeat the previous step for each of the variables in the **Variable** list. Close the dialog box by clicking **OK**.



In the next steps you specify expressions which simplify the definition of the constraint equations.

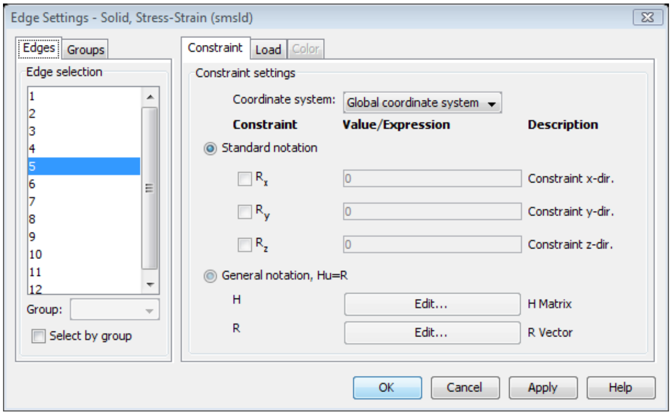
9 Select **Options>Expressions>Edge Expression**. In the **Edge Expressions** dialog box select Edge 5 and enter the following expressions in the **Name** and **Expression** columns. Click **OK**.

NAME	EXPRESSION
a21	x2-x1
b21	y2-y1
c21	z2-z1
xn	(x*a21+y*b21+z*c21)/sqrt(a21^2+b21^2+c21^2)
xn1	(x1*a21+y1*b21+z1*c21)/sqrt(a21^2+b21^2+c21^2)
xn2	(x2*a21+y2*b21+z2*c21)/sqrt(a21^2+b21^2+c21^2)
xn21	xn2-xn1
xn2n	xn2-xn
xnn1	xn-xn1
R1	-c21*(u1*xn2n+u2*xnn1)/xn21+a21*(w1*xn2n+w2*xnn1)/xn21
R2	-c21*(v1*xn2n+v2*xnn1)/xn21+b21*(w1*xn2n+w2*xnn1)/xn21
R3	-a21*(v1*xn2n+v2*xnn1)/xn21+b21*(u1*xn2n+u2*xnn1)/xn21



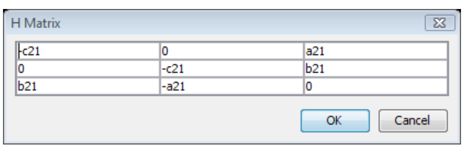
In the last step you set the constraints for the edge. In the graphical user interface you can use the general notation which allows you to specify a system of equation containing any linear combination of displacement components.

10 Open the **Edge Settings** dialog box by selecting **Physics>Edge Settings**. Select Edge 5 and select the **General Notation** check box.



11 Click the **Edit** button next to the **H** label and enter the following matrix:

$$\begin{bmatrix} -c21 & 0 & a21 \\ 0 & -c21 & b21 \\ b21 & -a21 & 0 \end{bmatrix}$$

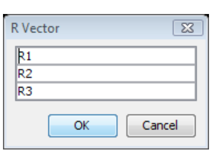


The H matrix multiplied by the displacement vector makes up the left hand side of the equation system, while the R vector which you enter next is the right hand side.

12 Click **OK**.

13 Click the **Edit** button next to the **R** label and enter the following vector:

$$\begin{bmatrix} R1 \\ R2 \\ R3 \end{bmatrix}$$



14 Click **OK**.

15 Click **OK** to close the **Edge Settings** dialog box.

16 In the **Boundary Settings** dialog box apply the **Fixed** constraint to Boundary 1, then click **OK**.

17 Click the **Solve** button on the Main toolbar to solve the problem.

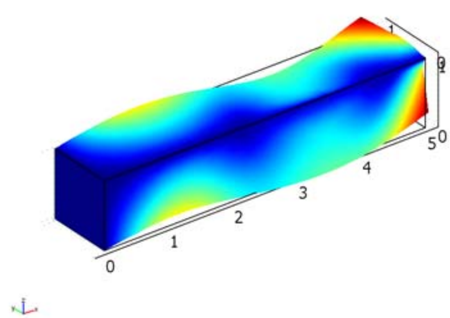
Next create a deformed shape plot of the bar. For better visualization of the straight edge plot the last eigenfrequency, which is 467.6 Hz.

18 Click the **Plot Parameters** button on the Main toolbar.

19 On the **General** page clear the **Slice** check box and select the **Boundary** and **Deformed shape** check boxes.

20 From the **Solution to use** list select the last eigenfrequency, which is 467.626 Hz.

21 Click **OK**.



EXAMPLE: STRAIGHT EDGE CONSTRAINT BY TRUSS ELEMENTS

An alternate method to that shown in the example on page 75 is to enforce the straight-edge constraint of a solid by adding a 3D Truss application mode to a model and activate it only on the edge to be constrained. Keep in mind that by adding truss elements to the model you add both additional mass and stiffness to the problem, which can influence the results.

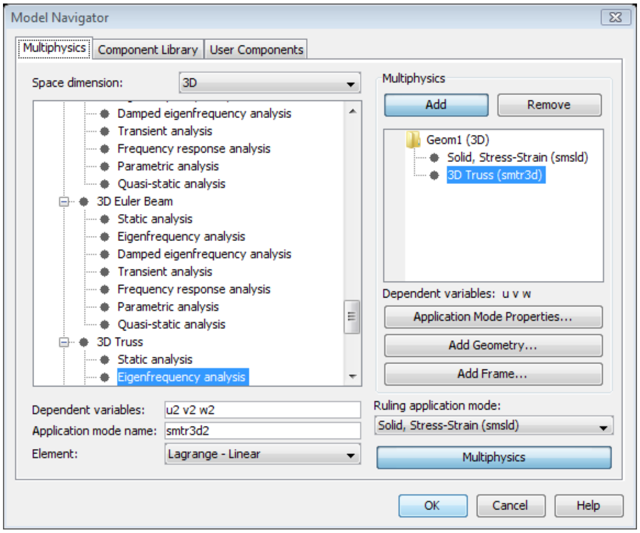
1 Repeat Steps 1 to 3 of the example on page 81.

2 In the **Boundary Settings** dialog box apply the **Fixed** constraint to Boundary 1, then click **OK**.

Continue by adding a truss application mode to the model.

3 Select **Multiphysics>Model Navigator**.

4 In the list of application modes select **Structural Mechanics Module>3D Truss>Eigenfrequency analysis**. Click **Add**, then click **OK**.

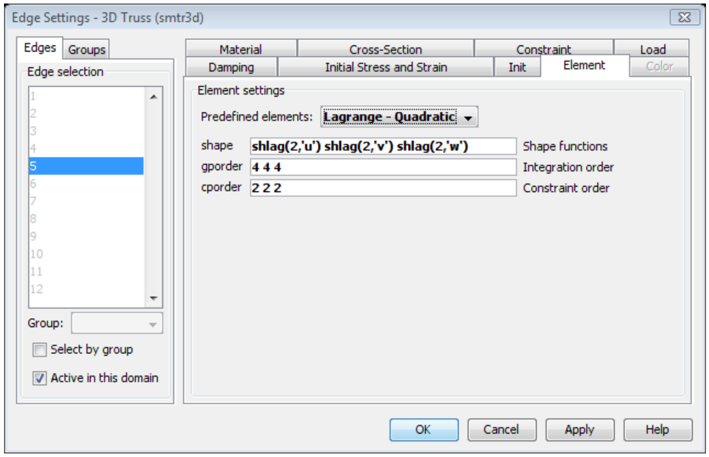


5 Select **Physics>Edge Settings** to open the **Edge Settings** dialog box for the truss application mode.

6 Select Edges 1–4 and 6–12, then clear the **Active in this domain** check box.

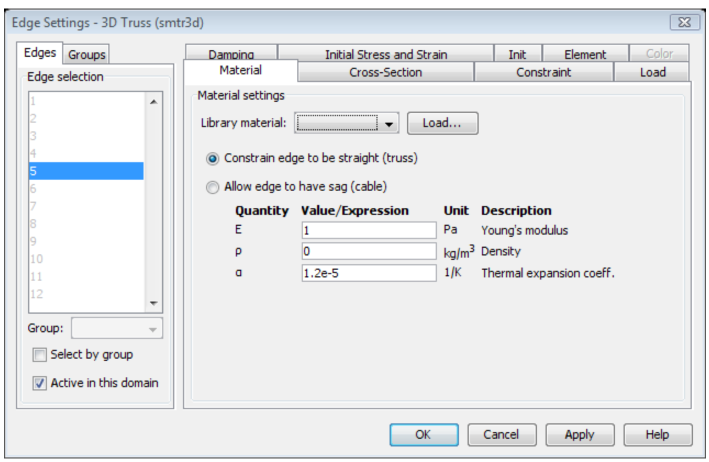
Next change the element type for the Truss application mode so that it is compatible with the default element type used in the Solid, Stress-Strain application mode.

7 Select Edge 5. On the **Element** page select **Lagrange - Quadratic** from the **Predefined elements** list.



Decrease the density and Young's modulus of the truss, so that it does not influence the results.

8 On the **Material** page, enter 1 in the **E** edit field and 0 in the ρ edit field. Click **OK**.



9 Click the **Solve** button on the Main toolbar to solve the problem.

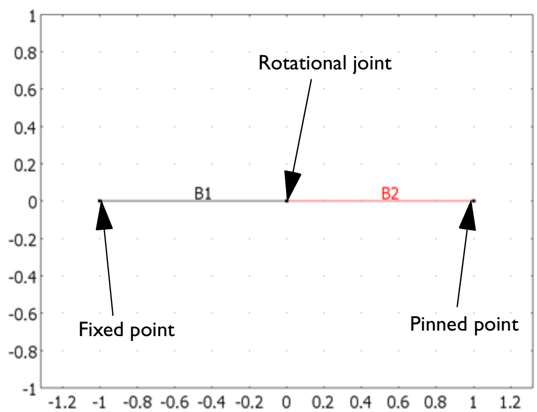
10 Repeat Steps 18–21 from the previous example to verify that the solution is similar.

Rotational Joints

Joints between elements in the In-Plane Truss and 3D Truss application modes are automatically rotational joints because the truss elements have no rotational degrees of freedom. In an application mode for beams, however, the rotational degrees of freedom are by default coupled between elements. To create a rotational joint between two beam elements, add one additional In-Plane Euler Beam or 3D Euler Beam application mode to your geometry. Make sure that it is only active for the boundary that includes the point where the joint will be positioned and that no other application mode is active here. Couple the translational degrees of freedom and leave the rotational degrees of freedom uncoupled at the joint. This procedure is described for a simple 2D case in the following example.

EXAMPLE: ROTATIONAL JOINT BETWEEN 2D BEAM ELEMENTS

In this example you set up a rotational joint by using two In-Plane Euler Beam application modes. You also use boundary expressions to gain access to dependent variables of both application modes during postprocessing.

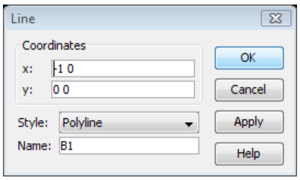


1 In the **Model Navigator** select **2D** from the **Space dimension** list.

2 In the list of application modes select **Structural Mechanics Module>In-Plane Euler Beam>Eigenfrequency analysis**; then click **OK**.

3 Hold down the Shift key and click the **Line** button on the Draw toolbar. Enter the following values for the line B1:

EDIT FIELD	B1
x	-1 0
y	0 0



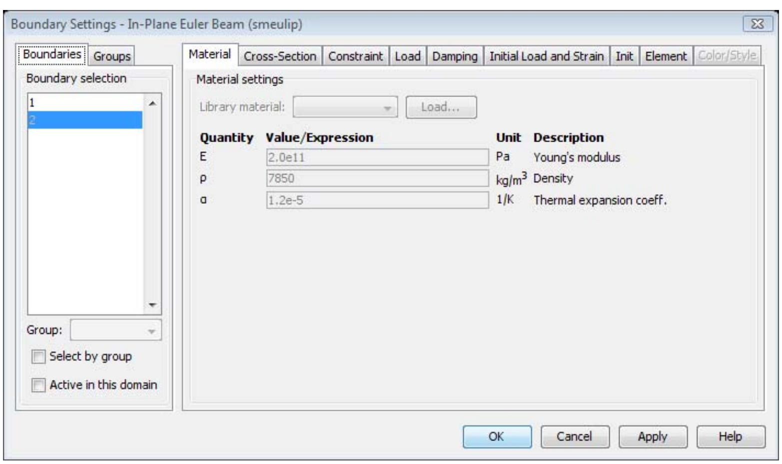
4 Click **OK**.

5 To draw the next line segment, hold down the Shift key and click the **Line** button on the Draw toolbar. Enter the following values for the line B2:

EDIT FIELD	B2
x	0 1
y	0 0

6 Click **OK**.

7 Select **Physics>Boundary Settings**. In the **Boundary selection** list select Boundary 2 and clear the **Active in this domain** check box, then click **OK**.



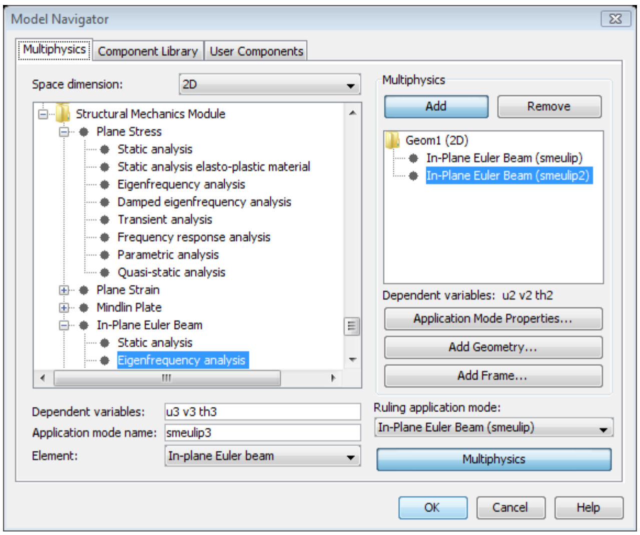
8 Select **Physics>Point Settings**. Select Point 1 from the **Point selection** list and on the **Constraint** page select **Fixed** from the **Constraint condition** list.

9 Click **OK**.

Continue by adding one more beam application mode to the model.

10 Select **Multiphysics>Model Navigator**.

11 In the list of application modes select **Structural Mechanics Module>In-Plane Euler Beam>Eigenfrequency analysis**. Click **Add**, then click **OK**.



Now you can edit the physics settings of the second application mode and couple the two together.

12 If the **Model Tree** is not visible click the **Model Tree** button on the Main toolbar to make it visible.

13 In the **Model Tree** click the **Detail** button or the **Inspect** button.

14 Expand the **In-Plane Euler Beam (smeulip2)** branch and double click **Boundary Settings** to open the **Boundary Settings** dialog box for this application mode.

15 In the **Boundary selection** list select Boundary 1 and clear the **Active in this domain** check box, then click **OK**.

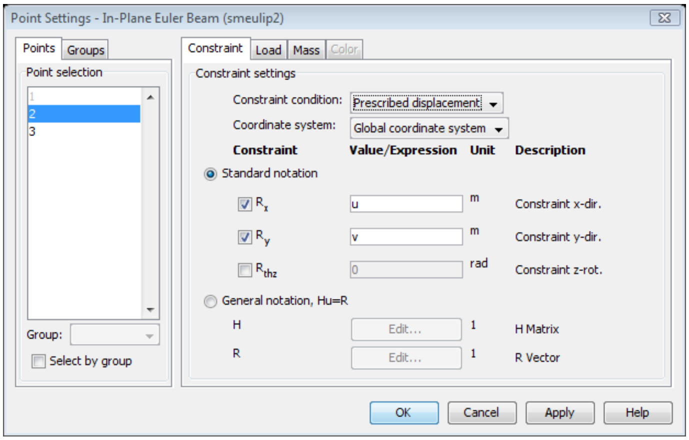
16 Under the **In-Plane Euler Beam (smeulip2)** branch double-click **Point Settings** to open the **Point Settings** dialog box for this application mode.

17 In the **Point selection** list select Point 3 and on the **Constraint** page select **Pinned** from the **Constraint condition** list.

18 Now select Point 2 and select **Prescribed displacement** from the **Constraint condition** list.

19 Select the $\mathbf{R_x}$ check box and enter u in the edit field next to it.

20 Select the $\mathbf{R_y}$ check box and enter v in the edit field next to it. Click **OK**.

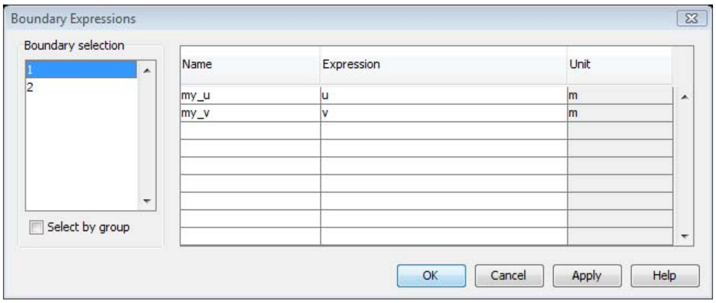


Now you can create boundary expressions that take the value of the displacement variables from both of the application modes and make them available on the entire geometry for postprocessing.

21 Select **Options>Expressions>Boundary Expressions**.

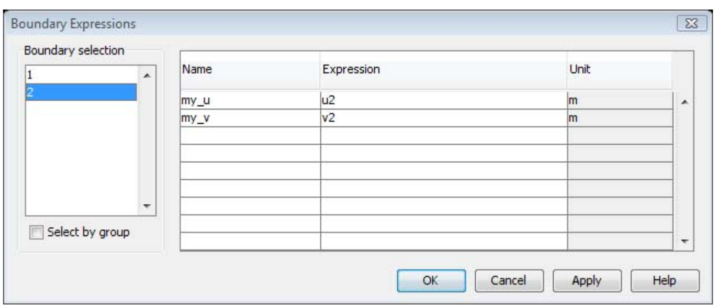
22 In **Boundary Expressions** dialog box select Boundary 1 and enter the following:

NAME	EXPRESSION
`my_u`	`u`
`my_v`	`v`



23 Select Boundary 2 and fill in the **Expression** column according to the table below:

NAME	EXPRESSION
`my_u`	`u2`
`my_v`	`v2`



24 Click **OK**.

25 Click the **Solve** button on the Main toolbar to solve the problem.

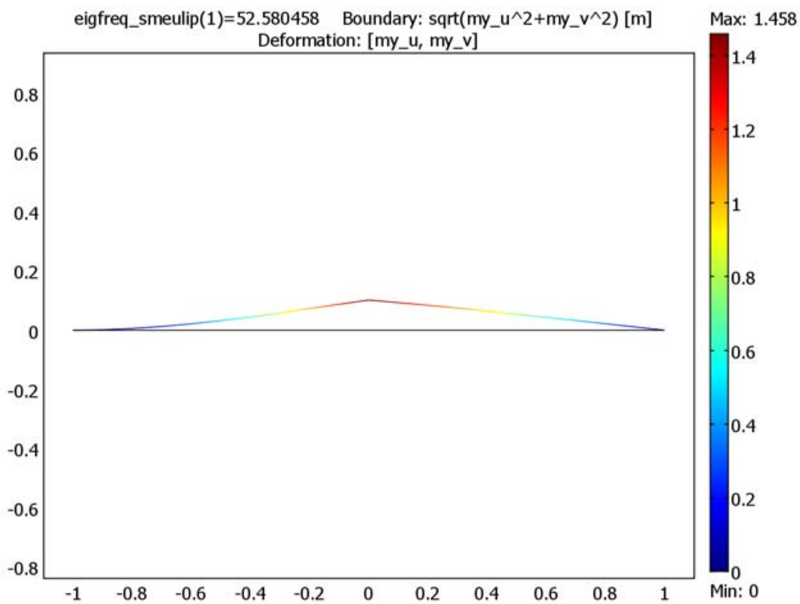
To see the deformation of the entire geometry you can use the boundary expressions you have defined.

26 Click the **Plot Parameters** button on the Main toolbar.

27 On the **General** page select the **Boundary** and **Deformed shape** check boxes.

28 On the **Deform** page, click the **Boundary Data** tab. Enter `my_u` in the **x component** edit field and `my_v` in the **y component** edit field.

29 On the **Boundary** page enter `sqrt(my_u^2+my_v^2)` in the **Expression** edit field. Click **OK** to plot the results.



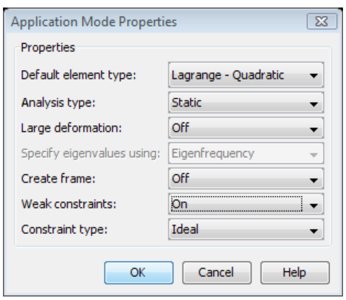
Reaction Forces

There are two possibilities for calculating the reaction forces on constrained boundaries in the Structural Mechanics Module. To get accurate results you can activate weak constraints which adds extra variables, corresponding to the reaction forces, to the solution components. You can also obtain approximate values without adding additional DOFs to the model by evaluating the surface traction on constrained boundaries.

Continuum Application Modes

CALCULATING BY USING WEAK CONSTRAINTS

To get the maximum accuracy for reaction forces, activate weak constraints for the application mode by selecting **On** in the **Weak Constraints** list box in the **Application Mode Properties** dialog box. Make sure that **Ideal** is selected for the **Constraint type**.



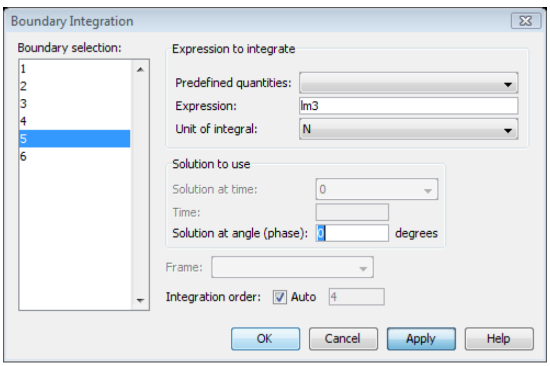
With weak constraints activated, COMSOL Multiphysics adds the reaction forces to the solution components. The variables in 3D are denoted `lm1`, `lm2`, and `lm3` (for Lagrange multipliers). Only the first two are present in 2D application modes. The following table shows the interpretation of these variables on the boundaries.

VARIABLE	CORRESPONDS TO REACTION FORCE IN THE DIRECTION OF THE FOLLOWING AXES					
	GLOBAL COORDINATE SYSTEMS			LOCAL GEOMETRICAL COORDINATE SYSTEMS		USER-DEFINED COORDINATE SYSTEMS
	3D	2D	2D AXI-SYMMETRIC	3D	2D	
lm1	x	x	r	t_1	t	x_1

VARIABLE	CORRESPONDS TO REACTION FORCE IN THE DIRECTION OF THE FOLLOWING AXES					
	GLOBAL COORDINATE SYSTEMS			LOCAL GEOMETRICAL COORDINATE SYSTEMS		USER-DEFINED COORDINATE SYSTEMS
	3D	2D	2D AXI-SYMMETRIC	3D	2D	
lm2	y	y	z	t_2	n	x_2
lm3	z	-	-	n	-	x_3

It is only possible to evaluate reaction forces on constrained boundaries in the constraint directions. Coordinate systems available in the Structural Mechanics Module are explained on page 144.

In order to calculate the reaction force on a boundary, you can carry out a boundary integration of one of the variables `lm1`, `lm2`, or `lm3` in the **Boundary Integration** dialog box. COMSOL Multiphysics displays the value of the integral in the message log.



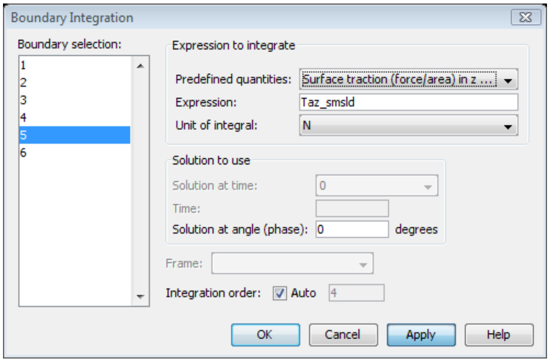
In a similar fashion use the **Subdomain Integration**, **Edge Integration**, or **Point Evaluation** dialog boxes to evaluate the reaction forces for constraints applied on these boundary types.

Because the reaction force variables are added to the solution components, the number of DOFs for the model increases slightly, depending on the mesh size for the boundaries in question.

With weak constraints activated, you are able to individually turn off weak constraints for boundaries on the **Weak Constr.** page of the **Boundary Settings** dialog box. However, boundaries that are adjacent to each other must have the same settings. The reason for this is that adjacent boundaries share a common node. Read more about the use of weak constraints on page 300 of the *COMSOL Multiphysics Modeling Guide*.

CALCULATING BY USING SURFACE TRACTION

By an alternative method, you can obtain an approximation to the reaction forces on constrained boundaries using boundary integration of the relevant components of the surface traction vector.



In 2D application modes, you need to multiply the surface traction by the cross section thickness before integrating to calculate the total reaction force. This method is less accurate than solving for the reaction forces by the use of weak constraints.

Other Application Modes

Evaluate the reaction forces or moments similarly as for the continuum application modes (see page 96). Use the following tables to identify the variables and corresponding forces or moments.

TRUSS APPLICATION MODES

VARIABLE	CORRESPONDS TO REACTION FORCE IN THE DIRECTION OF THE FOLLOWING AXES			
	GLOBAL COORDINATE SYSTEM		LOCAL GEOMETRICAL COORDINATE SYSTEM	USER-DEFINED COORDINATE SYSTEM
	3D	2D	2D	
Im1	x	x	t	x_1
Im2	y	y	n	x_2
Im3	z	-	-	x_3

BEAM APPLICATION MODES

VARIABLE	CORRESPONDS TO REACTION FORCE IN THE DIRECTION OF THE FOLLOWING AXES					
	GLOBAL COORDINATE SYSTEM		BEAM LOCAL COORDINATE SYSTEM	LOCAL GEOMETRICAL COORDINATE SYSTEM	USER-DEFINED COORDINATE SYSTEM	
	3D	2D	3D	2D	3D	2D
Im1	x	x	x_{local}	t	x_1	x_1
Im2	y	y	y_{local}	n	x_2	x_2
Im3	z	-	z_{local}	-	x_3	-

VARIABLE	CORRESPONDS TO REACTION MOMENT AROUND THE FOLLOWING AXES					
	GLOBAL COORDINATE SYSTEM		BEAM LOCAL COORDINATE SYSTEM	LOCAL GEOMETRICAL COORDINATE SYSTEM	USER-DEFINED COORDINATE SYSTEM	
	3D	2D	3D	2D	3D	2D
Im3	-	z	-	z	-	x_3
Im4	x	-	x_{local}	-	x_1	-
Im5	y	-	y_{local}	-	x_2	-
Im6	z	-	z_{local}	-	x_3	-

SHELL APPLICATION MODE

VARIABLE	CORRESPONDS TO REACTION FORCE IN THE DIRECTION OF THE FOLLOWING AXES			
	GLOBAL COORDINATE SYSTEM	SHELL LOCAL COORDINATE SYSTEM	LOCAL GEOMETRICAL COORDINATE SYSTEM	USER-DEFINED COORDINATE SYSTEM
lm1	x	x_{local}	t_1	x_1
lm2	y	y_{local}	t_2	x_2
lm3	z	z_{local}	n	x_3

VARIABLE	CORRESPONDS TO REACTION MOMENT AROUND THE FOLLOWING AXES			
	GLOBAL COORDINATE SYSTEM	SHELL LOCAL COORDINATE SYSTEM	LOCAL GEOMETRICAL COORDINATE SYSTEM	USER-DEFINED COORDINATE SYSTEM
lm4	x	x_{local}	t_1	x_1
lm5	y	y_{local}	t_2	x_2
lm6	z	z_{local}	n	x_3

MINDLIN PLATE APPLICATION MODE

VARIABLE	CORRESPONDS TO REACTION FORCE IN THE DIRECTION OF THE FOLLOWING AXES		
	GLOBAL COORDINATE SYSTEM	LOCAL GEOMETRICAL COORDINATE SYSTEM	USER-DEFINED COORDINATE SYSTEM
lm1	z	z	z

VARIABLE	CORRESPONDS TO REACTION MOMENT AROUND THE FOLLOWING AXES		
	GLOBAL COORDINATE SYSTEM	LOCAL GEOMETRICAL COORDINATE SYSTEM	USER-DEFINED COORDINATE SYSTEM
lm2	x	t	x_1
lm3	y	n	x_2

Material Models

A wide variety of material models is available in the Structural Mechanics Module. In addition, you can extend these models by modifying or defining your own material models. The chapters of this book dealing with the different application modes contain theory background and information about entering material settings in the graphical user interface, as well as application mode specific information. While all the material models are available in the continuum application modes, the other application modes use only linear elastic stress-strain relationship. Use the table below to locate the appropriate pages.

APPLICATION MODE	THEORY BACKGROUND	MATERIAL SETTINGS
Continuum Application Modes	page 166	page 193
Mindlin Plates	page 229	page 240
Shells	page 301	page 306
Beams	page 254	page 261
Trusses	page 280	page 287
Piezoelectric application modes	page 319	page 328

In the present section you find tips and tricks related to the use of material models in the Structural Mechanics Module.

Linear Elastic Materials

While for the isotropic case two parameters are enough to describe the material behavior, the number of parameters increases to (at most) 21 for the anisotropic case in 3D. When setting up a model make sure that the parameters you use are defined in agreement with the definitions used in the Structural Mechanics Module. The stress-strain relationship for linear elastic materials is discussed on page 167. If necessary, transform the material data before entering it in the user interface. For example, for orthotropic materials calculate the Poisson's ratio ν_{xy} by

$$\nu_{xy} = \nu_{yx}\frac{E_x}{E_y}.$$

Hyperelastic Materials

Two of the most widely used phenomenological models for hyperelastic materials: the Neo-Hookean and Mooney-Rivlin material models, are predefined in the Structural Mechanics Module. In addition, using predefined and custom stress and strain measures you can specify other material models. Read more about this on page 109.

Extracting the parameters for the material models from experimental stress strain curves involves curve fitting with the appropriate equations for the nominal (Piola-Kirchhoff) stress. These can be derived from the general stress strain relationship for hyperelastic materials, Equation 7-3, for a known strain state.

For a thin sheet of incompressible hyperelastic material under uniaxial tension the stretch along the axis of the loading is $\lambda_1 = \lambda = 1 + \varepsilon$, where ε is the strain. Due to symmetry and incompressibility the stretch ratios in the transverse directions are $\lambda_2 = \lambda_3 = \lambda^{-1/2}$. The stress state corresponding to this state of deformation is reduced to $P_1 = P$, $P_2 = P_3 = 0$, where

$$P = 2(\lambda - \lambda^{-2})\left(\frac{\partial W_{\mathrm{hyp}}}{\partial I_1} + \frac{1}{\lambda}\frac{\partial W_{\mathrm{hyp}}}{\partial I_2}\right). \qquad (4-2)$$

Assume a strain energy function, W_{hyp}, according to the Mooney-Rivlin material model, Equation 7-5, for which Equation 4-2 becomes

$$P = 2(\lambda - \lambda^{-2})C_{10} + 2(1 - \lambda^{-3})C_{01}. \qquad (4-3)$$

Use Equation 4-3 to curve fit the parameters C_{10} and C_{01} from a uniaxial stress-strain curve, as demonstrated by the following example script.

EXAMPLE: SCRIPT FOR CURVE FITTING OF PARAMETERS OF THE MOONEY-RIVLIN MATERIAL MODEL

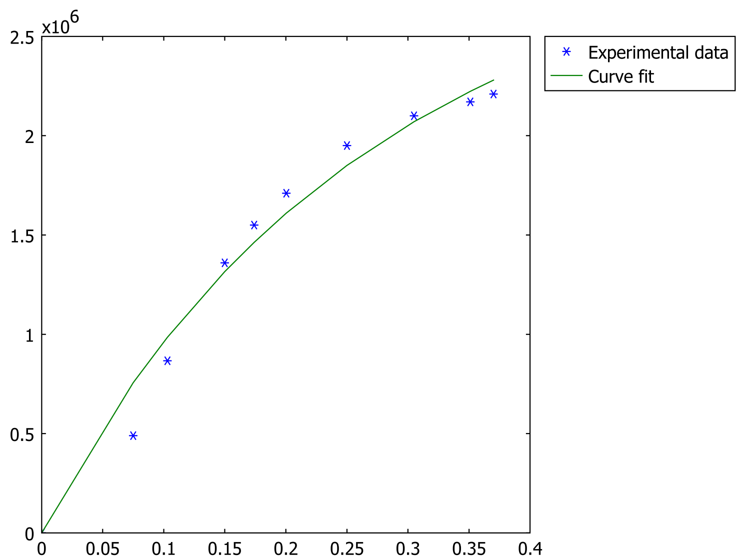
In the beginning of the script, stress and strain data are stored in the vectors engStressExp and engStrain, respectively. Then, the stretch, lam1, is calculated from the strain, followed by solving the equation for C_{10} and C_{01}. Both the experimental and fitted data are plotted at the end.

```
% Example
% Use least-squares analysis to curve fit C01 and C10 for a
% Mooney-Rivlin material from experimental stress strain data.
engStrain = [0, .075, .103, .15, .174, .2004, .25, .305, .351, .37];
engStressExp = [0, 4.9e5, 8.67e5, 1.36e6, 1.55e6, 1.71e6, 1.95e6,
2.10e6, 2.17e6, 2.21e6];
```

```
% Calculate stretch and set up equation system
lam1 = 1 + engStrain;
lam1=lam1';
CoeffMatrix = zeros(length(engStrain),2);
CoeffMatrix(:,1)  = 2./lam1.*(lam1.^2-1./lam1);
CoeffMatrix(:,2)  = 2./(lam1.^2).*(lam1.^2-1./lam1);

% Solve for C10 and C01
C10C01=CoeffMatrix\engStressExp';

% Calculate stress from this data
engStressMRML = CoeffMatrix*C10C01;
%Plot
plot(engStrain, engStressExp,'*', engStrain, engStressMRML, '-');
legend('Experimental data','Curve fit');
```



Elasto-Plastic Materials

The **Elasto-Plastic Material Settings** dialog box is prepared for directly entering three types of hardening models: perfectly plastic hardening, isotropic hardening, and kinematic hardening. You also have the choice to use von Mises or user-defined yield functions. The implementation of elasto-plastic material models in the Structural Mechanical Module works best for strains within the small strain range and permits large deformations.

To specify a hardening function for an elasto-plastic material model with isotropic hardening, enter it as an expression or use a function specified by a function table. In both cases the hardening function is a function of the effective plastic strain, ε_{pe}, and has to describe the behavior starting at the yield stress of the material. For the derivation of the hardening function, note that the experimental stress curve is a function of the total strain, which is the sum of the plastic strain and the elastic strain. Thus, the total effective strain can be written as

$$\varepsilon_{eff} = \varepsilon_{pe} + \frac{\sigma_e}{E} \tag{4-4}$$

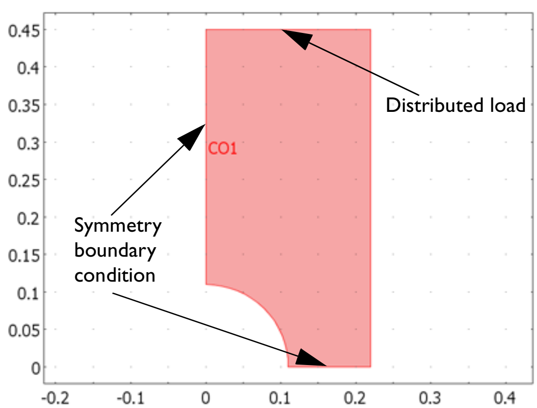
where σ_e is the effective stress and E is the Young's modulus for the material. Based on the experimental stress function, σ_{exp}, the hardening function can now be defined as

$$\sigma_{yhard} = \sigma_{exp}(\varepsilon_{eff}) - \sigma_{ys} = \sigma_{exp}\left(\varepsilon_{pe} + \frac{\sigma_e}{E}\right) - \sigma_{ys} \tag{4-5}$$

where σ_{ys} is the yield stress for the material.

EXAMPLE: USE OF STRESS-STRAIN CURVE AS HARDENING FUNCTION

The following steps show how to specify a hardening function based on a stress-strain curve from a uniaxial tension test. The model shows the loading under tension of a thin plate with a hole in the center. Due to symmetry analyze only a quarter of the plate.



1 In the **Model Navigator** select **2D** from the **Space dimension** list.

2 In the list of application modes select **Structural Mechanics Module>Plane Stress>Static analysis elasto-plastic material**; then click **OK**.

Create the geometry for the model. First, draw a rectangle, then a circle, and finally take the difference of these two objects to get the geometry for the plate.

3 Hold down the Shift key and click the **Rectangle/Square** button on the Draw toolbar. Enter `0.22` in the **Width** edit field and enter `0.45` in the **Height** edit field. Click **OK**.

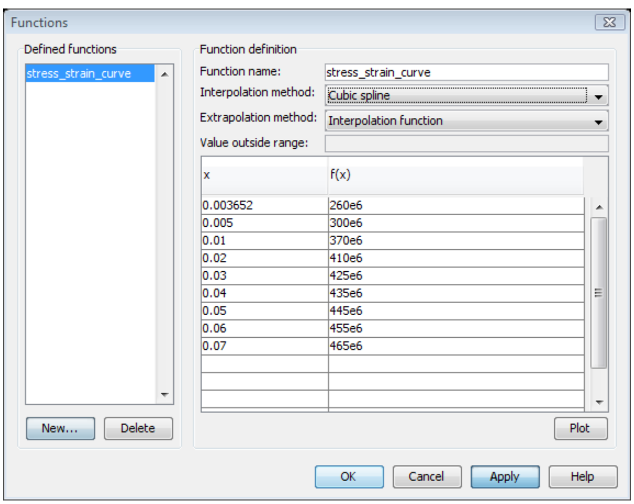
4 Hold down the Shift key and click the **Ellipse/Circle (Centered)** button on the Draw toolbar. In the **Radius** edit field enter `0.11`. Click **OK**.

5 With both objects selected click the **Difference** button on the Draw toolbar.

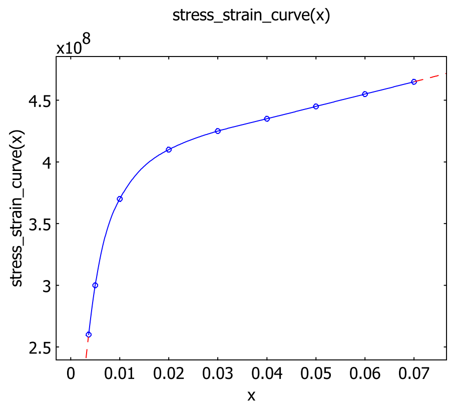
Continue by defining the experimental stress strain data as an interpolation function based on a table.

6 Select **Options>Functions**. Click **New** in the dialog box that opens. Enter `stress_strain_curve` in the **Function name** edit field and select the **Interpolation** radio button. Make sure that **Table** is selected in the **Use data from** list box and click **OK**.

7 In the **Functions** dialog box select **Cubic spline** from the **Interpolation** method and enter the data shown in the figure below in the **x** and **f(x)** columns.



8 Click the **Plot** button to see a plot of both the entered data points and the interpolated curve.



9 Click **OK** to close the **Functions** dialog box.

Next define the constants for the material parameters and the expression for the hardening function, based on the second part of Equation 4-5.

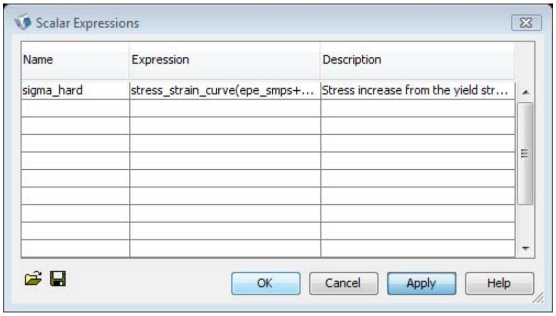
10 Select **Options>Constants** to open the **Constants** dialog box and enter the following constants; click **OK** when finished.

NAME	EXPRESSION	DESCRIPTION
E	71.2e9	Young's modulus
nu	0.31	Poisson's ratio
sigma_yield	260e6	Yield strength
Area	0.45*0.002	Area of cross section

11 Select **Options>Expressions>Scalar Expressions** and enter the expression from the following table. Click **OK** when finished.

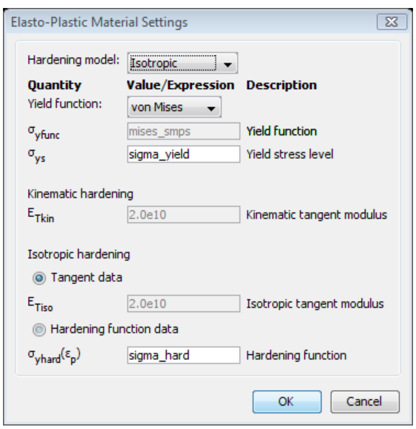
NAME	EXPRESSION	DESCRIPTION
sigma_hard	stress_strain_curve(epe_smps+mises_smps/E)-sigma_yield	Stress increase from the yield stress level due to hardening

The variables `epe_smps` and `mises_smps` are the effective plastic strain and the effective stress according to von Mises, respectively. The suffix changes depending on the application mode, in this case `_smps` denotes the Plane Stress application mode.



12 Select **Physics>Subdomain Settings**. In the **Subdomain selection** list select Subdomain 1 and enter `E` in the **E** edit field and `nu` in the ν edit field.

13 Click the **Elasto-Plastic Material Data** button to open the **Elasto-Plastic Material Settings** dialog box. Enter `sigma_yield` in the σ_{ys} edit field. Click the **Hardening function data** option button and enter `sigma_hard` in the $\sigma_{yhard}(\varepsilon_p)$ edit field.



Now enter the material parameters and boundary conditions for the model.

14 Click **OK** and then click **OK** again in the **Subdomain Settings** dialog box.

15 Select **Physics>Boundary Settings**. In the **Boundary selection** list select Boundaries 1 and 3. On the **Constraint** page select **Symmetry plane** from the **Constraint condition** list box.

16 Select Boundary 2 and on the **Load** page enter `Load/Area` in the $\mathbf{F_y}$ edit field. Click the **Edge load is defined as fore/area using the thickness** button; then click **OK**.

The last step is to define the load parameter and solve the problem.

17 Click the **Solver Parameters** button on the Main toolbar. Enter `Load` in the **Parameter name** edit field and enter `10e3:10e3:45e3 49e3 54e3 55e3:10e3:145e3 147e3:10e3:167e3` in the **Parameter list** edit field. Click **OK**.

18 Click the **Solve** button on the Main toolbar to solve the problem.

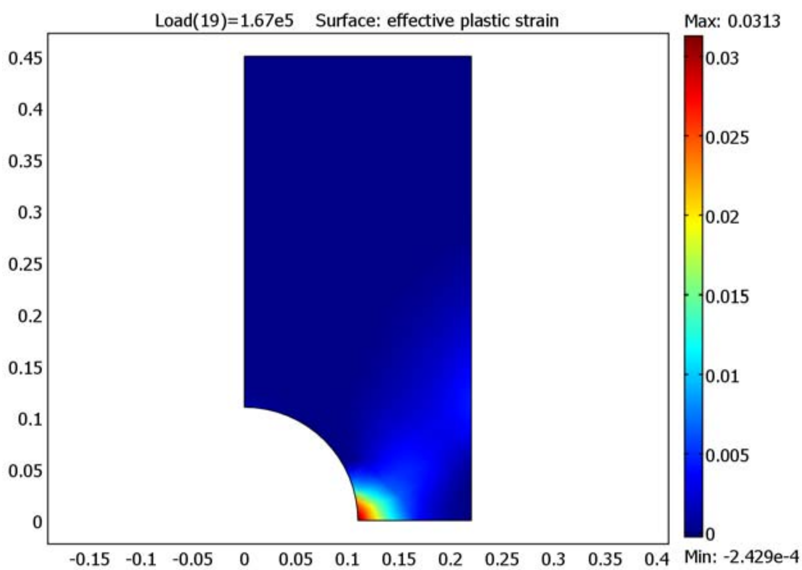
To visualize the solution you can plot the plastic strain.

19 Click the **Plot Parameters** toolbar button.

20 In the dialog box that opens switch to the **Surface** page.

21 From the **Predefined quantities** list, select **Plane Stress (smps)>effective plastic strain**.

22 Click **OK**.



Mixed Formulation

As described on page 170, the negative mean stress becomes an additional dependent variable when you select the **Use mixed U-P formulation** check box on the **Material** page of the **Subdomain Settings** dialog box. Select this setting when the Poisson's ratio of a material is close to 0.5, which means that the material is nearly incompressible. The mixed formulation is useful not only for elastic materials but also for elasto-plastic and hyperelastic materials.

Note that not all iterative solvers work together with mixed formulation because the stiffness matrix becomes indefinite. You can find recommendations on solver settings for the mixed formulation on page 126. It is also important to remember that because the shape function for the pressure must be one order less than the shape functions for the displacements, it is not possible to use linear elements for the displacement variables on the subdomains where mixed formulation is turned on.

User-Defined Materials

For most cases when a material model is not readily available in the graphical user interface, you can implement it by modifying or adding necessary equations and expressions.

For example, you can define a new hyperelastic material model by following these short steps of entering constants and variables and editing the strain energy function:

- Define the material parameters that you want to use in the strain energy function. Do this in the **Constants** dialog box that you open from the **Options** menu.
- Specify additional strain invariants to use in the strain energy expression, if applicable. Read about application mode variables in the *Structural Mechanics Module Reference Guide*. To include custom strain invariants in the strain energy function enter these as expressions. For example, choose **Options>Expressions>Scalar Expressions** to enter expressions that becomes available on the current geometry.
- Edit the definition of the strain energy function. Choose **Physics>Equation Systems>Subdomain Settings** and in the dialog box that opens click the **Variables** tab. Locate and edit the expression for the variable named `Ws_smsld`, which is the strain energy function. The index appended to the name `Ws` may be different depending on the application mode's name.

Example models in the *Structural Mechanics Module Model Library* provide further insight into how to use other types of constitutive equations in your models. Read about how to model the viscoelastic behavior of polymer material on page 443 and about modeling thermally induced creep in metals on page 479. These models provide good examples on setting up custom constitutive equations.

Material Libraries

A useful feature in COMSOL Multiphysics is the Materials/Coefficients library. In addition to the Basic Material Properties library the Structural Mechanics Module extends this library with two extra material libraries:

- MEMS Material Properties, an extended solid material library for MEMS applications. See “MEMS Material Properties Library” on page 111.
- Piezoelectric Material Properties, a material library with 23 common piezoelectric materials. See the section “Piezoelectric Material Properties Library” below.

The Basic Material Properties library is included with COMSOL Multiphysics and contains properties for a limited number of basic solid materials, given as constants, and temperature-dependent properties for air and water, given as functions.

For more information about using the **Materials/Coefficients Library** dialog box, see “Using the Materials/Coefficients Library” on page 223 in the *COMSOL Multiphysics User’s Guide*.

Piezoelectric Material Properties Library

The Piezoelectric Material Properties library ships with the Acoustics Module, MEMS Module, and Structural Mechanics Module. It contains the following piezoelectric materials:

MATERIAL
Barium Sodium Niobate
Barium Titanate
Barium Titanate (poled)
Lithium Niobate
Lithium Tantalate
Lead Zirconate Titanate (PZT-2)
Lead Zirconate Titanate (PZT-4)
Lead Zirconate Titanate (PZT-4D)
Lead Zirconate Titanate (PZT-5A)
Lead Zirconate Titanate (PZT-5H)
Lead Zirconate Titanate (PZT-5J)

MATERIAL
Lead Zirconate Titanate (PZT-7A)
Lead Zirconate Titanate (PZT-8)
Quartz
Rochelle Salt
Bismuth Germanate
Cadmium Sulfide
Gallium Arsenide
Tellurium Dioxide
Zinc Oxide
Zinc Sulfide
Ammonium Dihydrogen Phosphate
Aluminum Nitride

All materials define the following material properties needed for piezoelectric modeling:

MATERIAL PROPERTY	DESCRIPTION
c_E	Elasticity matrix
e	Coupling matrix, stress-charge
ε_{rS}	Relative permittivity, stress-charge
s_E	Compliance matrix
d	Coupling matrix, strain-charge
ε_{rT}	Relative permittivity, strain-charge
ρ	Density

MEMS Material Properties Library

The MEMS Material Properties library ships with the Acoustics Module, MEMS Module, and Structural Mechanics Module. It contains 33 materials commonly used in MEMS applications. The materials are divided into the following groups: Metals, Semiconductors, Insulators, and Polymers.

The basic structure of this library comes from the book *Microsensors, MEMS, and Smart Devices* (Ref. 3). The material properties come from two primary sources: the *CRC Handbook of Chemistry and Physics* (Ref. 1) and *MacMillan's Chemical and Physical Data* (Ref. 2). Some of the mechanical properties in the library are instead more MEMS-specific values from *The MEMS Handbook* (Ref. 4), and most of the semiconductor properties are values from Ref. 5. Ref. 6 provides a valuable resource for cross-checking the insulation material properties.

The table below lists the materials and their corresponding groups:

MATERIAL	GROUP
Aluminium (Al)	Metals
Silver (Ag)	Metals
Gold (Au)	Metals
Chrome (Cr)	Metals
Indium (In)	Metals
Titanium (Ti)	Metals
Iron (Fe)	Metals
Nickel (Ni)	Metals
Lead (Pb)	Metals
Palladium (Pd)	Metals
Platine (Pt)	Metals
Antimon (Sb)	Metals
Tungsten (W)	Metals
C [100]	Semiconductors
GaAs	Semiconductors
Ge	Semiconductors
InSb	Semiconductors
Si(c)	Semiconductors
Poly-Si	Semiconductors
Silicon (single-crystal)	Semiconductors
Al2O3	Insulators
SiC (6H)	Insulators
Si3N4	Insulators
SiO2	Insulators
ZnO	Insulators
Borosilicate	Insulators
Nylon	Polymers
PMMA	Polymers

MATERIAL	GROUP
Polymide	Polymers
Polyethylene	Polymers
PTFE	Polymers
PVC	Polymers

REFERENCES

1. D.R. Lide (Editor-in-chief), *CRC Handbook of Chemistry and Physics*, 84th edition, CRC Press, 2003.

2. A.M. James and M.P. Lord, *MacMillan's Chemical and Physical Data*, MacMillan's Press, 1992.

3. J.W. Gardner, V.K. Varadan, and O.O. Awadelkarim, *Microsensors, MEMS, and Smart Devices*, John Wiley & Sons, 2001.

4. M. Gad-el-Hak (editor), *The MEMS Handbook*, CRC Press, 2002.

5. *New Semiconductor Materials. Characteristics and Properties*, http://www.ioffe.ru/SVA/NSM, 2003.

6. *Ceramics WebBook*, http://www.ceramics.nist.gov/srd/scd/scdquery.htm, 2003.

Multiphysics Modeling

In this section you find modeling tips about how to create multiphysics models with the Structural Mechanics Module. A good place to start reading about how to create all types of multiphysics models is “Multiphysics Modeling” on page 63 of the *COMSOL Multiphysics Modeling Guide*, where you can see how to add or remove different physics in a model, how to set the properties for the different physics and how to manage the solution components.

Thermal-Structure Interaction

The Structural Mechanics Module provides a predefined one-way coupling for thermal-structure interaction, which combines a continuum application mode from the Structural Mechanics Module with a heat transfer application mode from the Heat Transfer Module or COMSOL Multiphysics. See “Thermal-Structure Interaction” on page 354.

You can also manually set up such a coupling using the temperature described by a heat transfer application mode to define the strain temperature. On the **Load** page in the **Subdomain Settings** dialog box in the structural mechanics application mode, select the **Include thermal expansion** check box, and enter the dependent variable for temperature from the heat transfer application mode, typically T, in the **Temp** edit field for the strain temperature.

Note: A special approach is required if the structural analysis is performed in the frequency domain. This includes the following analysis types: **Frequency response**, **Eigenfrequency**, and **Damped eigenfrequency**. The coupled displacement-temperature field presents thermoelastic oscillations of small amplitude, which are initialized to zero. You need to set the strain reference temperature **Tempref** to zero, and use a special form of the heat balance equation. For more details, see the model “Heat Generation in a Vibrating Structure” on page 703 of the *Structural Mechanics Module Model Library.*

By default, COMSOL Multiphysics solves for the temperature and displacements simultaneously. For large problems, you can take advantage of the one-way coupling and solve the problem sequentially (unless there are thermal properties that depend on the displacements): first solve for temperature and then perform the stress-strain analysis using the computed temperature field from the heat transfer equation. While the matrices from a coupled thermal-structure model are unsymmetric, the individual structural and heat transfer problems can result in symmetric matrices. The SPOOLES solver takes advantage of matrix symmetry and further reduces memory requirements. See "Solving for a Subset of the Dependent Variables" on page 329 of the *COMSOL Multiphysics Modeling Guide* on how to select which variables to solve for.

See Chapter 15, "Thermal-Structure Interaction," in the *Structural Mechanics Module Model Library* for models that exemplify thermal-structure interaction.

Fluid-Structure Interaction

Fluid-structure interaction models usually include a two-way coupling between the solid and fluid domains. The fluid exerts a force on the solid domain, while the deformation of the solid affects the geometry of the fluid domain. The Fluid-Structure Interaction (FSI) predefined multiphysics coupling enables this interaction by combining fluid flow with structural mechanics and using a Moving Mesh (ALE) application mode. You can find a description of this multiphysics coupling on page 356 of this book. See Chapter 10, "Fluid-Structure Interaction," in the *Structural Mechanics Module Model Library* for models that exemplify thermal-structure interaction.

A first thing to consider is whether your FSI problem really needs two-way coupling between the physics. If you do not expect the deformation of the solid domain to influence the flow problem, the ALE application mode is not necessary. This way you can greatly simplify your model. A good example where this method is applied is the model "Fluid-Structure Interaction in a Network of Blood Vessels" on page 210 of the *Structural Mechanics Module Model Library.*

When using the Moving Mesh (ALE) application mode it can happen that, as the mesh is deforming, you get inverted elements, which result in convergence problems. There are several ways to avoid this:

- Try to start with a different mesh. It is often preferable to start from a reasonably uniform mesh.
- Try the remeshing algorithm. This allows you to create a new mesh before the one you are working with becomes too distorted.

- Try a different smoothing algorithm. Winslow smoothing is slightly slower, more memory consuming, and is usually, but not always, better than Laplace smoothing.
- Another technique you can try is to draw help lines along which you can control the mesh deformation.

You find a more detailed description of the above techniques in the COMSOL Support Knowledge Base; see Ref. 1.

If you are using the transient solver with a coupled model and you experience convergence problems for the initial time step it is most likely due to the instantaneous application of a boundary condition, like the velocity. To help the problem to converge at the initial time step you can use a smoothed step transition function, like `flc1hs` (a smoothed Heaviside function), when defining the boundary condition. This way both the velocity and its derivative are zero for $t = 0$. You can read about using smoothed step functions in the COMSOL Support Knowledge Base; see Ref. 2.

Acoustic-Structure Interaction

By coupling application modes from the Structural Mechanics Module to an acoustics application mode from either the Acoustics Module or COMSOL Multiphysics you can solve acoustic-structure interaction. Coupling to an acoustics application mode from COMSOL Multiphysics enables you to analyze the sound field in an interior space. By using an acoustics application mode from the Acoustics Module you can have additional tools like a transient solver and means to simulate absorbing or radiation boundary conditions. You can find models, including step by step instructions, on acoustic-structure interaction in Chapter 2, "Acoustic-Structure Interaction," of the Structural Mechanics Module Model Library.

In the multiphysics coupling, on the boundaries between the solid and fluid, the acoustic analysis provides a load (the sound pressure) to the structural analysis, and the structural analysis provides accelerations to the acoustic analysis. For example, the structural mechanics application mode in a 3D model is the **Solid, Stress-Strain (smsld)** application mode. On the **Load** page of the **Boundary Settings** dialog box for this application mode specify `-p*nx_smsld`, `-p*ny_smsld`, and `-p*nz_smsld` in the edit fields labeled $\mathbf{F_x}$, $\mathbf{F_y}$, and $\mathbf{F_z}$, respectively. The variables `nx_smsld`, `ny_smsld`, and `nz_smsld` are the Cartesian components of the normal vector directed outward from the subdomain where the **Solid, Stress-Strain (smsld)** application mode is active. In the **Boundary Settings** dialog box for the acoustics application mode select the **Conditions** page and set the boundary condition to **Normal acceleration** for those same boundaries. Specify the acceleration, $\mathbf{a_n}$, as `nx_smsld*u_tt_smsld+ny_smsld*v_tt_smsld+nz_smsld*w_tt_smsld`. The variables `u_tt_smsld`, `v_tt_smsld`, and `w_tt_smsld` are the acceleration components from the **Solid, Stress-Strain (smsld)** application mode. These and other application mode variables are available on the **Variables** page of the dialog box that opens if you go to **Physics>Equation System>Boundary Settings**.

Coupled acoustic-structure models have symmetric matrices, which means you can take advantage of the SPOOLES solver to reduce memory requirements.

When coupling acoustics time-harmonic and structural mechanics frequency response application modes make sure that the excitation frequencies for these are set to the same value. You can find these variables in the **Application Scalar Variables** dialog box, which opens if you select **Physics>Scalar Variables**. One more thing to look out for is the mesh size where the acoustics application modes are active. A good rule of thumb is that the mesh should have about 10–12 elements per wavelength.

References

1. COMSOL Support Knowledge Base, Solution no. 970, http://www.comsol.com/support.

2. COMSOL Support Knowledge Base, Solution no. 905, http://www.comsol.com/support.

Contact Modeling

In the Structural Mechanics Module you can create models involving contact, with or without friction, between parts. Contact is implemented based on the *augmented Lagrangian* method, which is described on page 186. When modeling contact between structural parts you need to set up *contact pairs*, which define where the parts may come into contact. A contact pair consists of two sets of boundaries, which are the *master domains* and the *slave domains*. The 2D and 3D structural continuum application modes use the pairs to set up equations that prevent the *slave boundaries* to penetrate the *master boundaries*. The present section provides some advice regarding important aspects of creating contact models. You can find tips regarding solver settings for contact models in the section "Solver Settings for Contact Modeling" on page 129.

On page 217 you can read about how to specify contact pairs and define the physics for these in the graphical user interface.

When creating contact models it can often be to advantage to set up a prototype in 2D before attempting a 3D model. Similarly it is often good to start using linear elements to ease convergence toward a solution. When you have got this working, you can switch to quadratic elements if you want to.

You can find contact models complete with step by step instructions in Chapter 7, "Contact and Friction Models," of the *Structural Mechanics Module Model Library*.

Constraints

Make sure that the bodies are sufficiently constrained, also in the initial position. If the bodies are not in contact in the initial configuration, and there are no constraints on the bodies, you have an underconstrained state. This causes the solver to fail. One way to fix this problem is to set initial values for the displacement variables so that you have a small penetration in the initial configuration. Another way is to use a displacement-controlled model rather than a force-controlled one.

Contact Pairs

For efficiency, only include those boundaries that may actually come in contact in the slave. For the master, it is often a bit more efficient to make it so large that every slave point “has” a corresponding master point. Note that the corresponding master point is obtained by following the normal to the slave until it reaches the master.

To decide which boundaries should be assigned as master and slave in a contact pair consider the following guidelines:

- Make sure that the master boundary stiffness in the normal direction is higher than the slave boundary stiffness. This is especially important if the difference in stiffness is quite large, for example, over ten times larger. Keep also in mind that for elasto-plastic or hyperelastic materials there can be a significant change in stiffness during the solution process, and choose the master and slave boundaries accordingly. For such materials you might have to also adjust the penalty factor as the solution progresses.
- When the contacting parts have approximately the same stiffness, you can instead consider the geometry of the boundaries. The master should be concave and the slave convex rather than the opposite.

Once you have chosen the master and slave boundaries you should mesh the slave finer than the master. Do not make the slave mesh just barely finer than the master because this often causes unphysical oscillations in the contact pressure. Make the slave at least two times finer than the master.

Boundary Settings for Contact Pairs

PENALTY FACTORS

Note that in the augmented Lagrangian method, the value of the penalty factor does not affect the accuracy of the final solution, like it does in the penalty method. When running into convergence problems, check the penalty parameters. If the iteration process fails in some of the first augmented iterations, lower your penalty parameters. If the model seems to converge but very slowly, consider increasing the maximum value of your penalty parameters.

Increasing the penalty factor can lead to an ill-conditioned Jacobian matrix and convergence problems in the Newton iterations. You can often see this by noting that the damping factor becomes less than 1 for many Newton iterations. If this occurs, decrease the penalty factors.

The default values for the penalty factors, using Young’s modulus, only work for linear isotropic materials, for which the Young’s modulus is defined. For other types of materials you need to substitute $\boldsymbol{E}$ with a suitable value or define it as a constant or expression variable. For elasto-plastic materials you may find that the default value works fine until there is a significant decrease in stiffness due to plastic deformation. This can give rise to convergence problems for the nonlinear solver, since the penalty factor becomes too large. To aid convergence you can specify an expression for the stiffness that depends for example on the solver parameter.

INITIAL VALUE

In force-controlled contact problems where no other stiffness prohibits the deformation except the contact, the initial contact pressure is crucial for convergence. If it is too low the parts might pass through each other in the first iteration. If it is too high they never come into contact.

Multiphysics Contact

Multiphysics contact problems are often very ill-conditioned, which leads to convergence problems for the nonlinear solver. For example, take heat transfer through the contact area, where initially only one point is in contact. The solution for the temperature is extremely sensitive to the size of the contact area (that is, the problem to determine the temperature is ill-conditioned). Therefore it is important to resolve the size of the contact area accurately, that is, to use a very fine mesh in the contact area. If the contact area is larger, you do not need as fine mesh because then the temperature solution is not that sensitive to the size of the contact area. If possible, start with an initial configuration where the contact area is not very small.

Damping

Damping is important in time-dependent and frequency response analysis. This section describes how to model it in the Structural Mechanics Module using different damping models.

Rayleigh Damping

A common model for viscous damping is *Rayleigh damping*, which assumes that the damping is proportional to a linear combination of the stiffness and mass. To illustrate this, consider a system with a single degree of freedom. The following equation of motion describes the dynamics of such a system with viscous damping:

$$m\frac{d^2u}{dt} + c\frac{du}{dt} + ku = f(t) .$$

In the Rayleigh damping model, the damping parameter c is expressed in terms of the mass m and the stiffness k as

$$c = \alpha_{dM}m + \beta_{dK}k$$

where α_{dM} and β_{dK} are the mass and stiffness damping parameters, respectively.

A complication with the Rayleigh damping model is obtaining good values for the damping parameters. A more physical damping measure is the damping ratio, the ratio between actual and critical damping, often expressed as a damping factor in percentage of the critical damping. You can find commonly used values of damping factors in the literature.

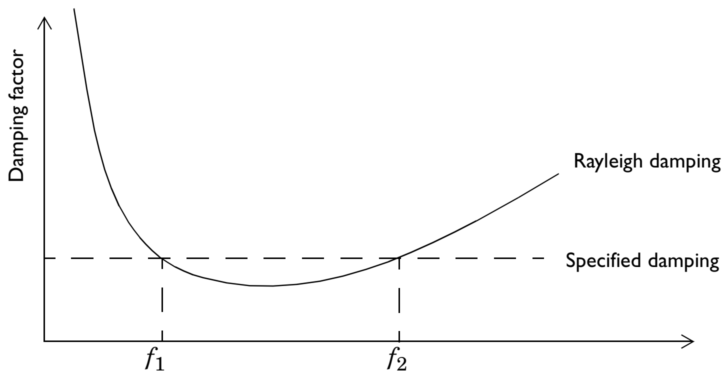
It is possible to transform damping factors to Rayleigh damping parameters. The damping factor, ξ, for a specified pairs of Rayleigh parameters, α_{dM} and β_{dK}, at a frequency, f, is

$$\xi = \frac{1}{2}\left(\frac{\alpha_{dM}}{2\pi f} + \beta_{dK}2\pi f\right) .$$

Using this relationship at two frequencies, f_1 and f_2, with different damping factors, ξ_1 and ξ_2, results in an equation system that can be solved for α_{dM} and β_{dK}:

$$\begin{bmatrix} \frac{1}{4\pi f_1} & \pi f_1 \\ \frac{1}{4\pi f_2} & \pi f_2 \end{bmatrix} \begin{bmatrix} \alpha_{dM} \\ \beta_{dK} \end{bmatrix} = \begin{bmatrix} \xi_1 \\ \xi_2 \end{bmatrix}.$$

Using the same damping factors, $\xi_1 = \xi_2$, does not result in a constant damping factor inside the interval $f_1 < f < f_2$. It can be shown that the damping factor is lower inside the interval, as the following figure shows.



Note: All application modes in the Structural Mechanics Modules use nonzero default values for α_{dM} and β_{dK}. You must adapt these default values to suit your specific modeling situation.

Loss Factor Damping

Loss factor damping (sometimes referred to as material or structural damping) applies to viscoelastic materials modeled in the frequency domain. The complex modulus $G^*(\omega)$ is the frequency-domain representation of the stress relaxation function of viscoelastic material. It is defined as

$$G^* = G' + jG'' = (1 + j\eta)G'$$

where G' is the storage modulus, G'' is the loss modulus, and their ratio $\eta = G''/G'$ is the *loss factor*. The term G' defines the amount of stored energy for the applied strain, whereas G'' defines the amount of energy dissipated as heat; G', G'', and η can all be frequency dependent.

In COMSOL Multiphysics, the loss information appears as a multiplier of the total strain in the stress-strain relationship:

$$\sigma = D((1+j\eta)\varepsilon - \varepsilon_{\mathrm{th}} - \varepsilon_0) + \sigma_0\,.$$

For hyperelastic material, the loss information appears as a multiplier in the first Piola-Kirchhoff stress, P:

$$P = (1+j\eta)\frac{\partial W_{\mathrm{hyp}}}{\partial \nabla \mathbf{u}}$$

Loss factor damping is available for frequency response analysis in all application modes, but it is not defined for elasto-plastic materials.

Equivalent Viscous Damping

Although equivalent viscous damping is independent of frequency, it is only possible to use it in a frequency response analysis. Equivalent viscous damping also uses a loss factor η as the damping parameter, but its implementation is different from the actual loss factor damping.

The piezoelectric application modes have built-in support for this type of damping. For the other application modes, you can model it using the stiffness damping parameter β_{dK}. Specify β_{dK} to the loss factor, η, divided by the excitation frequency.

$$\beta_{dK} = \frac{\eta}{2\pi f} = \frac{\eta}{\omega}$$

You must also set the mass damping factor, α_{dM}, to zero.

Explicit Damping

Another way to model damping is to specify it explicitly as a viscous force. In a transient analysis you do so by specifying a force that depends on the velocities with opposite signs:

$$\mathbf{F} = -c\mathbf{v}$$

where **v** is the velocity

$$\mathbf{v} = \begin{bmatrix} u_t \\ v_t \\ w_t \end{bmatrix}$$

and u_t is the velocity component in the x direction, typically named `ut`.

You can specify viscous damping locally using such a force on any domain level.

In a frequency response analysis you can specify viscous damping in a similar way, but the name of the velocity variable changes and includes the application mode name, for example, `u_t_smsld` for the Solid, Stress-Strain application mode.

No Damping

To create an undamped model, you can also select to use **No damping** from the **Damping model** list.

Fatigue Analysis

The fatigue analysis capabilities in the Structural Mechanics Module extends the COMSOL Script environment—an open and extensible language for technical computing of any kind—with a suite of tools for performing fatigue analysis.

The fatigue analysis tools works together with both MATLAB and COMSOL Script.

The fatigue analysis capabilities in the Structural Mechanics Module consists of a number of script functions that compute fatigue damage or fatigue life from input consisting of loading data and material fatigue data. A typical function takes all stress tensor components in the form of a matrix as input and delivers the fatigue damage or fatigue life as result.

A typical fatigue analysis consists of the following steps:

1. Perform a finite element analysis of your structure, using COMSOL Multiphysics and the Structural Mechanics Module.
2. Calculate the stress field on a matrix format, available in COMSOL Script or MATLAB.
3. Calculate the fatigue damage from the stress field, and the fatigue material data using a fatigue-analysis script function.
4. Plot the result.

Fatigue analysis is divided into high-cycle and low-cycle fatigue depending of the number of load cycles. The Structural Mechanics Module as delivered can handle the following cases for both high-cycle and low-cycle fatigue:

- Proportional loading, constant amplitude
- Nonproportional loading, constant amplitude
- Proportional loading, nonconstant amplitude

You find a detailed description of how to perform fatigue analysis together with a comprehensive background to the subject and a theoretical descriptions of the different methods in the chapter “Fatigue Analysis” on page 361. Model examples are available in the section “Fatigue Models” on page 345 in the *Structural Mechanics Module Model Library.*

Solver Settings

A large number of possible solver settings are available in COMSOL Multiphysics. To make it easier for you to select a solver and its associated solver parameters, the various application modes use different default settings depending on the analysis type. In some situations you must change the default settings. This section helps you select a solver and its solver settings to solve structural mechanics and multiphysics problems. Further details about all solver settings appear in the chapter “Solving the Model” on page 359 in the *COMSOL Multiphysics User’s Guide*.

Symmetric Matrices

The **Matrix symmetry** list appears on the **General** page in the **Solver Parameters** dialog box. Here you specify if the assembled matrices (stiffness matrix, mass matrix) resulting from your equations are symmetric or not.

Normally the matrices from a single-physics structural mechanics problem are symmetric, but there are exceptions:

- Multiphysics models solving for several physics simultaneously, for example, heat transfer and structural mechanics. Solving for several structural mechanics application modes, such as shells combined with beams, does not create unsymmetric matrices.
- Elasto-plastic analysis.

One of the benefits of using the symmetric solvers is that they use less memory and are faster. The default option is **Automatic**, which means the solver automatically detects if the system is symmetric or not. Some solvers do not support symmetric matrices and always solve the full system regardless of symmetry. The default solver in 2D, UMFPACK, does not support symmetry—but it is faster than SPOOLES, the default solver in 3D. SPOOLES uses less memory, but memory is usually not a major issue in 2D.

Note: Selecting the **Symmetric** option for a model with unsymmetric matrices produces incorrect results.

Complex matrices can be unsymmetric, symmetric, or Hermitian. Hermitian matrices do not appear in structural mechanics problems.

Note: Selecting the **Hermitian** option for a model with complex-valued symmetric matrices produces incorrect results.

Selecting Iterative Solvers

The **Linear system solver** list appears on the **General** page in the **Solver Parameters** dialog box. The default solver in the Structural Mechanics Module is **Direct (SPOOLES)** in 3D and **Direct (UMFPACK)** in 2D. For large problems (several hundred thousands or millions of degrees of freedom) it is beneficial to use iterative solvers when possible to save time and memory. The drawback is that they are more sensitive and might not converge if the mesh quality is low.

The iterative solvers have more options than the direct solvers. The following table makes suggestions on which iterative solver and preconditioner to use for different analyses for large problems.

ANALYSIS	LINEAR SYSTEM SOLVER	PRECONDITIONER
Static analysis, single physics	Conjugate gradients	Geometric multigrid
Quasi-static transient analysis, single physics	Conjugate gradients	Geometric multigrid
Parametric analysis, single physics	Conjugate gradients	Geometric multigrid
Eigenfrequency analysis, single physics	Conjugate gradients	Geometric multigrid
Static analysis, multiphysics	GMRES	Geometric multigrid
Eigenfrequency analysis, multiphysics	GMRES	Geometric multigrid
Frequency response analysis	GMRES	Geometric multigrid
Elasto-plastic analysis	GMRES	Geometric multigrid
Time-dependent analysis	Conjugate gradients	Geometric multigrid

Specifying a positive shift greater than the lowest eigenfrequency results in indefinite matrices. The conjugate gradients iterative solver does not work for indefinite matrices. Get more details about solver settings in Chapter 6, “Selecting a Solver,” in the *COMSOL Multiphysics User’s Guide*.

Note: Check the mesh quality when using the geometric multigrid (GMG) preconditioner. It does not work well when using the option to scale the geometry before meshing (on the **Advanced** tab in the **Free Mesh Parameters** dialog box). When using extruded meshes, you might need to create the mesh cases manually.

The conjugate gradients solver does not work together with a mixed formulation because it results in an indefinite stiffness matrix. For this type of problems the following solver combinations work:

LINEAR SYSTEM SOLVER	PRECONDITIONER	SMOOTHER
GMRES	GMG	Vanka
GMG	-	Vanka
GMRES	Incomplete LU	-

When using the Vanka smoother for a mixed-formulation problem, specify the pressure as the Vanka variable. Get more information about using the Vanka smoother in the section “The Vanka Algorithm” on page 530 in the *COMSOL Multiphysics Reference Guide*.

Specifying the Absolute Tolerance

The absolute-tolerance parameters used for time-dependent problems are very problem specific. As a rule of thumb, set the absolute tolerance to be at least one order of magnitude smaller than the typical displacement.

The default value is 0.001 for all solution components. When solving mixed problems with both displacements and pressure, this default results in very small tolerance conditions for the pressure. One way to help the solver is to specify individual tolerance values for all solution components. This speeds up the solution and usually does not affect the accuracy. For example, when solving a model using the 3D Solid, Stress-Strain application mode for a mixed problem with a typical displacement amplitude of 10^{-5} and an internal pressure amplitude of 10^5, specify `u 1e-7 v 1e-7 w 1e-7 p 1e3` in the **Absolute tolerance** edit field (that is, use space-separated pairs of variable names and the absolute tolerance for that variable).

Solver Settings for Contact Modeling

You solve contact problems using the augmented Lagrangian method. The augmented solution components are specified on the **Stationary** page in the **Solver Parameters** dialog box. The augmented solution components are the contact pressure and the friction traction components. By default the solver finds these components automatically.

If the model includes friction, some solution components from the previous solution step are needed. You specify these variables on the **Parametric** page in the **Solver Parameters** dialog box. The components are the master coordinates, the contact variable, and, if dynamic friction is modeled, the time. By default the program finds these components automatically.

MANUAL SCALING

You need to use manual scaling if the parts are not in contact initially (initial value of contact pressure is zero) or if the model includes friction. Select **Manual** from the **Type of scaling** list on the **Advanced** page in the **Solver Parameters** dialog box. In the **Manual scaling** edit field, enter the name of all the solution components together with their approximate order of magnitude. For example, solving a plane stress problem with one contact pair including friction, where the displacements in both directions are around 10^{-3}, the contact pressure is around 1000, and the friction traction components are around 100. Then enter `u 1e-3 v 1e-3 Tn_cp1_smps 1000 Ttx_cp1_smps 100 Tty_cp1_smps 100` (using space-separated pairs of variable names and scaling factors) in the **Manual scaling** edit field.

To get the list of degrees of freedom in the model, go to the **Solver Manager** dialog box and look at the **Solve For** page. For each degree of freedom, use a positive value that is of the order of the typical value of that variable. You need not specify scaling factors for the friction history variables containing `_old`, for instance `contact_cp1_old_smps`, `xm_old_cp1_smps`, `ym_old_cp1_smps`.

You can read more about how to prevent ill-conditioned matrices by scaling of variables and equations on page 497 of the *COMSOL Multiphysics Reference Guide*.

TOLERANCES

You find tolerance settings for both the augmented Lagrangian solver and the nonlinear solver on the **Stationary** page of the **Solver Parameters** dialog box.

Specify the tolerance for the augmented Lagrangian solver (`augtol`) in the **Tolerance** edit field under the **Augmented Lagrangian solver** group label. It controls the accuracy of the so-called augmentation components T_n, T_{tx}, T_{ty} (that is, the contact pressure and the friction tractions). The accuracy in these components is the product of the manual scaling value with augtol. For example, if the manual scale for T_n is set to 10^8, the default `augtol` = 10^{-3} gives an error $10^8 * 10^{-3} = 10^5$ or about 0.1% in T_n.

The tolerance for the nonlinear solver (`ntol`) controls the accuracy of the displacement variables (and other variables in a multiphysics model). You can change its value in the **Relative Tolerance** edit field in the **Nonlinear settings** area.

Do not use a too coarse `ntol`, especially if the body is stiff, because this causes too large errors in the determination of the contact tractions, which leads to nonconvergence in the augmented Lagrangian iterations. You can estimate `ntol` by looking at the scaling of the dependent variables and the penalty factors:

$$\mathrm{ntol} < \frac{T_{\mathrm{min}} \cdot \mathrm{augtol}}{p_{\mathrm{max}} \cdot u_{\mathrm{max}}}$$

where T_{min} denotes the minimum of the contact traction scales, p_{max} the maximum penalty factor, and u_{max} the maximum of the displacement scale factors. For example, for a material with Young's modulus of 10^{11}, a minimum mesh size of 10^{-2}, and with the manual scaling set to

```
u 1e-4 v 1e-3 Tn_cp1_smps 1e8 Ttx_cp1_smps 1e6 Tty_cp1_smps 1e6
```

using the default values for the penalty factors, the nonlinear tolerance is

$$\mathrm{ntol} < \frac{10^6 \cdot 10^{-3}}{10^{13} \cdot 10^{-3}} = 10^{-7}$$

AUGMENTED LAGRANGIAN SOLVER

You select the augmented Lagrangian solver from the **Solver** list on the **Stationary** page of the **Solver Parameters** dialog box. This solver controls the updating of the contact tractions in each augmented Lagrange iteration. Because these degrees of freedom are rather few there is no performance issue here. The default lumped solver is used for 2D problems because this gives less undershoots in the contact tractions at the ends of the segments in contact. The lumped solver is an approximation that replaces the boundary mass matrix with a lumped diagonal matrix.

In 3D, the UMFPACK solver is used as default because lumping does not work for quadratic elements.

5

Application Mode Guide

The application modes in the Structural Mechanics Module form a complete set of simulation tools for various modeling situations in structural and solids mechanics. Select an application mode that describes your structure by analyzing the loading conditions and any possible engineering assumptions.

Overview

The following table lists the application modes available in the Structural Mechanics Module. For a detailed description of any of them, refer to the corresponding section on the page listed in the table.

The column for the dependent variables shows the field variables that formulate the PDEs or weak form equations. Depending on the engineering assumptions, these variables might be a subset of the displacements u, v, and w in the global coordinate system, or the rotations ϕ_x , ϕ_y , and ϕ_z about the global axes. In the piezoelectric application modes the electric potential V is included. For axisymmetric simulations, COMSOL Multiphysics uses a variable transformation to avoid a singularity at the symmetry axis.

For each application mode, the table indicates the availability of various analysis capabilities.

Finally the table lists the domains where you can specify application mode data such as material properties, loads, and constraints. Note that edges exist only in 3D geometries.

APPLICATION MODE		**PAGE**		**ANALYSIS CAPABILITIES**										**DOMAINS**			
	DEFAULT NAME	**DESCRIPTION**	**DEPENDENT VARIABLES**	**STATIC**	**EIGENFREQUENCY**	**TIME DEPENDENT**	**FREQUENCY RESPONSE**	**PARAMETRIC**	**QUASI-STATIC TRANSIENT**	**LARGE DEFORMATION**	**LINEAR BUCKLING**	**ELASTO-PLASTIC MATERIAL**	**BUILT IN TEMPERATURE COUPLING**	**POINT**	**EDGE**	**BOUNDARY**	**SUBDOMAIN**
CONTINUUM APPLICATION MODES		159															
Solid, Stress-Strain	smsld	160	u, v, w	√	√	√	√	√	√	√	√	√	√	√	√	√	√
Plane Stress	smps	161	u, v	√	√	√	√	√	√	√	√	√	√	√		√	√
Plane Strain	smpn	162	u, v	√	√	√	√	√	√	√	√	√	√	√		√	√
Axial Symmetry Stress-Strain	smaxi	163	uor, v	√	√	√	√	√	√	√	√	√	√	√		√	√

APPLICATION MODE		PAGE		ANALYSIS CAPABILITIES										DOMAINS			
	DEFAULT NAME	DESCRIPTION	DEPENDENT VARIABLES	STATIC	EIGENFREQUENCY	TIME DEPENDENT	FREQUENCY RESPONSE	PARAMETRIC	QUASI-STATIC TRANSIENT	LARGE DEFORMATION	LINEAR BUCKLING	ELASTO-PLASTIC MATERIAL	BUILT IN TEMPERATURE COUPLING	POINT	EDGE	BOUNDARY	SUBDOMAIN
MINDLIN PLATE	smdrm	227	w, ϕ_x, ϕ_y	√	√	√	√	√	√				√	√		√	√
BEAMS		253															
In-plane Euler Beam	smeulip	277	u, v, ϕ	√	√	√	√	√	√				√	√		√	
3D Euler Beam	smeul3d	278	$u, v, w, \phi_x, \phi_y, \phi_z$	√	√	√	√	√	√				√	√	√		
TRUSSES		279															
2D Truss	smtr2d	299	u, v	√	√	√	√	√	√	√	√		√	√		√	
3D Truss	smtr3d	300	u, v, w	√	√	√	√	√	√	√	√		√	√	√		
SHELL	smsh	304	$u, v, w, \phi_x, \phi_y, \phi_z$	√	√	√	√	√	√				√	√	√	√	
PIEZO APPLICATION MODES		319															
Piezo Solid	smpz3d	348	u, v, w, V	√	√	√	√	√						√	√	√	√
Piezo Plane Stress	smpps	348	u, v, V	√	√	√	√	√						√		√	√
Piezo Plane Strain	smppn	349	u, v, V	√	√	√	√	√						√		√	√
Piezo Axial Symmetry	smpaxi	349	uor, w, V	√	√	√	√	√						√		√	√

To change the type of simulation for a given set of parameters, simply modify the analysis type, which is an application mode property. The analysis type sets up the coefficients in the underlying equations. The available analysis types depend on the application mode. Static, eigenfrequency, time dependent, and frequency response are common for all application modes.

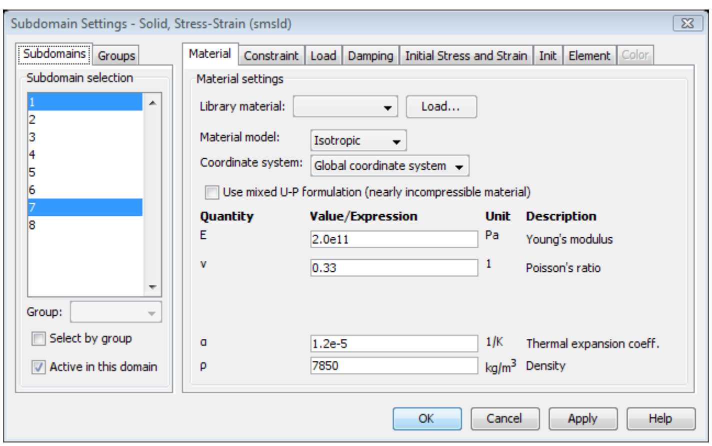
When starting to work on a model, select the application mode from the **Model Navigator**. You can add application modes to an existing model to create a multiphysics model. One example is adding Euler beams to a thin plate modeled in the Plane Stress application mode to account for various stiffening structures in the plate.

When using the Axial Symmetry application mode it is important to note that the horizontal axis represents the r direction and the vertical axis the z direction. The entire geometry must lie in the half plane $r > 0$.

Next note that the excitation frequency `freq` used in frequency response analysis is given as an *application scalar variable* in the GUI.

Depending on the application mode, you can specify parameters defining a problem on points, edges (3D), boundaries, and subdomains. It is possible to specify loads and constraints on all available domain types, but you can specify material properties only for the subdomain, except for shells and in-plane Euler beams, where they are defined on the boundary, and 3D Euler beams, where they are defined on the edge level.

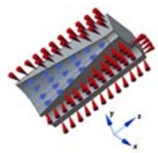
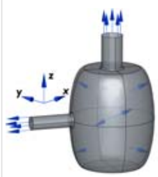
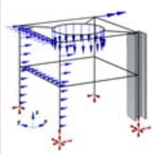
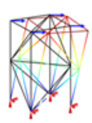
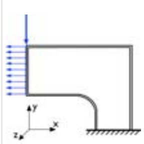
All domain-setting dialog boxes—**Point Setting**, **Edge Settings**, **Boundary Settings**, and **Subdomain Settings**—have a common layout. In each of these dialog boxes, you specify a problem using the tabbed pages **Constraints**, **Loads**, **Material**, **Cross Section**, **Init**, and **Element**. The **Material**, **Init**, and **Element** pages are available only on the subdomain level, except for shells and in-plane Euler beam, where they exists on the boundary, and 3D Euler beam, where they exists on edge level. In contrast, the **Load** and **Constraint** pages exist on all available domains, making it possible to define constraints and loads on all levels. You set the loads and constraints independently of each other, so it is possible to apply loads on constrained domains. Such loads do not affect the computation’s final result.

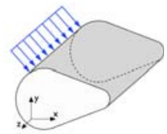
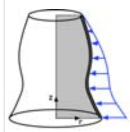
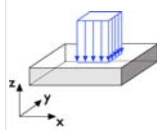
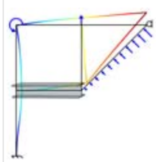
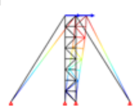
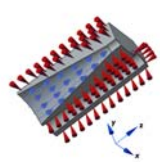


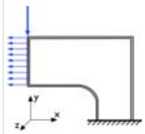
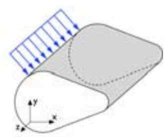
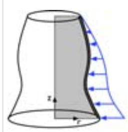
The **Cross Section** page is available only for the beam application modes. On the **Postprocessing** page you can indicate at which depth you want to postprocess results for plate and shell elements.

Selecting the Correct Application Mode

The Structural Mechanics Module supplies the following application modes:

APPLICATION MODE	PICTURE	USE TO MODEL
Solid, Stress-Strain		3D solids, not thin or slender.
Shell		Thin 3D structures.
3D Euler Beam		Slender 3D structures. Typical examples are frameworks and latticeworks.
3D Truss		Slender 3D structures with components capable to withstand axial forces only. Typical example is latticeworks.
Plane Stress		In-plane loaded thin plates.

APPLICATION MODE	PICTURE	USE TO MODEL
Plane Strain		In-plane loaded structures whose extent out of the plane is large compared to the in-plane dimensions, or when the z-displacement is in some way restricted. A typical example is a long tunnel.
Axial Symmetry Stress-Strain		Axisymmetric structures exposed to symmetric loads and constraints.
Mindlin Plate		Out-of-plane loaded thin plates.
2D Euler Beam		Slender 2D structures. Typical examples are plane frameworks and latticeworks
2D Truss		Slender 2D structures with components capable to withstand axial forces only. Typical example is plane latticeworks.
Piezo Solid		3D solids, of piezoelectric material

APPLICATION MODE	PICTURE	USE TO MODEL
Piezo Plane Stress		In-plane loaded thin plates of piezoelectric material
Piezo Plane Strain		In-plane loaded structures of piezoelectric material whose extent out of the plane is large compared to the in-plane dimensions, or when the z-displacement is in some way restricted.
Piezo Axial Symmetry		Axisymmetric structures of piezoelectric material exposed to symmetric loads and constraints.

The following chapters present a detailed description of the above-mentioned application modes together with an introductory example for each one. For a description of the command-line use of the application modes, see the corresponding entries in the section "Application Mode Programming Reference" on page 100.

Analysis Capabilities

The Structural Mechanics Module performs static, eigenfrequency, damped eigenfrequency, transient, frequency response, parametric, and quasi-static analyses. The analysis types require different solvers and equations. In the **Application Mode Properties** dialog box you select an analysis type, each of which has a predefined solver. You can disable the choice of a predefined solver by clearing the **Auto select solver** check box in the **Solver Parameters** dialog box. The following table lists the different analysis types with their predefined solver:

ANALYSIS TYPE	SOLVER
Static	Stationary
Static elasto-plastic material	Parametric
Eigenfrequency	Eigenvalue
Damped eigenfrequency	Eigenvalue

ANALYSIS TYPE	SOLVER
Transient	Time dependent
Frequency response	Parametric
Quasi-static	Time dependent
Linear buckling	Eigenvalue

To manually change to a different solver, make a new selection in the **Solver Parameters** dialog box. Read through the following solver descriptions to help find good candidates for your application.

STATIC ANALYSIS

A static analysis solves for stationary displacements, rotations, and temperature (depending on the type of application mode). All loads and constraints are constant. The equations include no mass or mass moment of inertia.

EIGENFREQUENCY ANALYSIS

An eigenfrequency analysis solves for the undamped eigenfrequencies and the shape of the eigenmodes. When performing an eigenfrequency analysis, you can specify whether to look at the mathematically more fundamental eigenvalue, λ, or the eigenfrequency, f, which is more commonly used in a structural mechanics context.

$$f = -\frac{\mathrm{Im}(\lambda)}{2\pi}$$

You control the way to specify eigenvalues from the **Application Mode Properties** dialog box from the **Physics** menu.

DAMPED EIGENFREQUENCY ANALYSIS

A damped eigenfrequency analysis solves for the damped eigenfrequencies and the shape of the eigenmodes. When performing a damped eigenfrequency analysis, you can specify whether to look at the mathematically more fundamental eigenvalue, λ, or the eigenfrequency, f, which is more commonly used in a structural mechanics context.

$$f = -\frac{\mathrm{Im}(\lambda)}{2\pi}$$

You control the way to specify eigenvalues from the **Application Mode Properties** dialog box (choose **Properties** from the **Physics** menu).

In addition to the eigenfrequency you can also look at the quality factor, Q, and decay factor, δ, of your model.

$$Q = \frac{\mathrm{Im}(\lambda)}{2\mathrm{Re}(\lambda)}$$

$$\delta = \mathrm{Re}(\lambda)$$

TRANSIENT ANALYSIS

A transient analysis solves a time-dependent (unsteady) problem where loads and constraints can vary in time.

For transient analysis, COMSOL Multiphysics models damping with the Rayleigh damping model, which assumes that the damping matrix $\boldsymbol{C}$ is a linear combination of the stiffness matrix $\boldsymbol{K}$ and the mass matrix $\boldsymbol{M}$:

$$C = \alpha_{dM}M + \beta_{dK}K$$

You can specify the Rayleigh damping parameters locally.

FREQUENCY RESPONSE ANALYSIS

A frequency response analysis solves for the steady-state response from harmonic loads. For this analysis type, you can model damping using Rayleigh damping (in the same way as in a transient analysis) or using loss factor damping, where you specify a loss factor.

For a frequency response analysis, the Structural Mechanics Module divides harmonic loads into two parts:

- The amplitude, $\boldsymbol{F}$
- The phase ($\boldsymbol{F}_{\mathrm{Ph}}$)

Together they define a harmonic load whose amplitude and phase shift can depend on the excitation angular frequency ω or excitation frequency f.

$$F_{\mathrm{freq}} = F(\omega) \cdot \cos\left(\omega t + F_{\mathrm{Ph}}(\omega) \cdot \frac{\pi}{180}\right)$$
$$\omega = 2\pi f$$

For a frequency response analysis, you can choose either the stationary linear or nonlinear solvers, setting the excitation frequency in the **Application Scalar Variables** dialog box. An easier way to perform a frequency sweep is to choose the parametric solver with `freq` as the named parameter. You set the sweeping frequency in the **List of parameter values** edit field, which appears on the **Parametric** page in the **Solver Parameters** dialog box.

The result of a frequency response analysis is a complex time-dependent displacement field, which can be interpreted as an amplitude u_{amp} and a phase angle u_{phase}. The actual displacement at any point in time is the real part of the solution:

$$u = u_{\mathrm{amp}}\cos(2\pi f \cdot t + u_{\mathrm{phase}})$$

COMSOL Multiphysics allows the visualization of the amplitudes and phases as well as the solution at a specific angle (time). The **Solution at angle** parameter makes this task easy. When plotting the solution, the program multiplies it by $e^{i\phi}$, where ϕ is the angle in radians that corresponds to the angle (specified in degrees) in the **Solution at angle** edit field. COMSOL Multiphysics plots the real part of the evaluated expression:

$$u = u_{\mathrm{amp}}\cos(\phi + u_{\mathrm{phase}})$$

The angle ϕ is available as the variable `phase` (in radians) and is allowed in plotting expressions. Both `freq` and `omega` are available variables.

Note: In a frequency response analysis, everything is treated as harmonic: prescribed displacements, velocities, accelerations, thermal strains, and initial stress and strains; not only the forces.

QUASI-STATIC ANALYSIS

A quasi-static transient analysis neglects mass effects, assuming the time scale in the structural mechanics problem is much smaller than other dynamics. An example is a transient thermal problem where the time scale in the thermal problem is often much longer than that of the structural dynamics.

PARAMETRIC ANALYSIS

A parametric analysis finds the solution dependence from the variation of a specific parameter. The parameter could be, for instance, a material property or the position of a load. The equations are static.

LARGE DEFORMATIONS

The Structural Mechanics Module allows you to include large deformations with the restriction of small strains in all fully dimensional application modes. This effect is also sometimes referred to as a nonlinear geometric effect. Using large deformation, the application mode replaces the normal strain with the Green strain and replaces the stress with the second Piola-Kirchhoff stress. It solves the problem using a *total Lagrangian formulation*.

LINEAR BUCKLING ANALYSIS

A linear buckling analysis includes the stiffening effects from stresses coming from nonlinear strain terms. The two stiffnesses coming from stresses and material define an eigenvalue problem where the eigenvalue is a load factor that, when multiplied with the actual load, gives the critical load in a linear context. The linear buckling analysis uses the eigenvalue solver.

Another way to calculate the critical load is to include large deformation effects and increase the load until the solver fails because the load has reached its critical value.

Linear buckling analysis is available only in the continuum and Truss application modes.

THERMAL COUPLINGS

Solids expand with temperature, which causes thermal strains to develop in the material. These thermal strains combine with the elastic strains from structural loads to form the total strain:

$$\varepsilon = \varepsilon_{\mathrm{el}} + \varepsilon_{\mathrm{th}}$$

Thermal strain depends on the temperature, T, the stress-free reference temperature, T_{ref}, and the thermal-expansion coefficient, α:

$$\varepsilon_{\mathrm{th}} = \alpha(T - T_{\mathrm{ref}})$$

Thermal expansion affects displacements, stresses, and strains. Thermal coupling is available as an option in all application modes except the piezoelectric application modes. You need only specify the thermal expansion coefficient and the two temperature fields, T and T_{ref}. These temperatures can be any mathematical expression and are typically other variables solved for in another COMSOL Multiphysics application mode, for instance, the heat transfer application modes. You can use temperature coupling in any type of analysis.

Note: A special approach is required if the structural analysis is performed in the frequency domain. This includes the following analysis types: frequency response, eigenfrequency, and damped eigenfrequency. The coupled displacement-temperature field presents thermoelastic oscillations of small amplitude, which are initialized to zero. You need to set the strain reference temperature **Tempref** to zero and use a special form of the heat balance equation. For more details, see the example “Heat Generation in a Vibrating Structure” on page 703 of the *Structural Mechanics Module Model Library.*

6

Coordinate Systems and Symbols

The Structural Mechanics Module makes available various predefined and user-defined coordinate systems, which are described in this chapter. In a separate section, you also find information about the symbols used for illustrating loads and constraints.

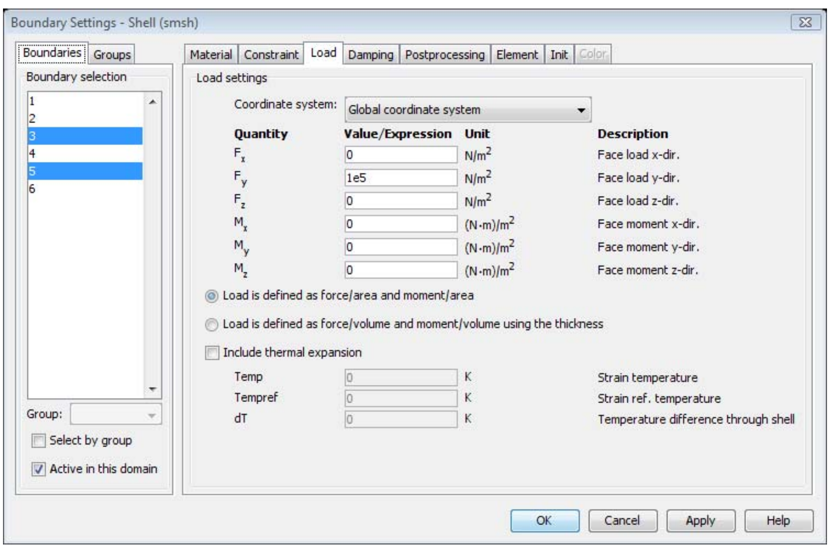
Coordinate Systems

Using different coordinate systems can be convenient when specifying loads, constraints, and anisotropic materials, and when postprocessing the results. The Structural Mechanics Module provides a number of different coordinate systems:

- A global Cartesian coordinate system, where the geometry is created in 3D (x, y, z).
- A local geometrical coordinate system on 2D boundaries (t, n) and on 3D faces (t_1, t_2, n).
- Application-mode specific coordinate systems: a shell coordinate system and a 3D Euler beam coordinate system.
- User-defined coordinate systems.

To specify the coordinate system, select it from the **Coordinate system** list on the **Constraint**, **Load**, and **Material** pages.

The following figure shows the **Load** page in the **Boundary Settings** dialog box for the Shell application mode.



The Global Coordinate System

You can use the global coordinate system in all application modes to specify loads and constraints on all domain levels: points, edges, faces, and subdomains. It is the default setting for loads and constraints in all application modes on all domain levels except boundary constraints for the Mindlin plate application mode. The default names for the space coordinates are the following for the different geometries:

GEOMETRY	DEFAULT NAME OF SPACE COORDINATES
2D	$x\ y\ z$
3D	$x\ y\ z$
Axial symmetry 2D	$r\ \varphi\ z$

It is possible to change the names of the space coordinates when creating a geometry from the **Model navigator**, see "Creating Cartesian and Cylindrical Coordinate Systems" on page 27 in *COMSOL Multiphysics User's Guide* for details.

Local Geometrical Coordinate Systems

Boundaries in 2D and 3D have geometric variables describing the parametrization of the geometry defined on them. These variables contain directions that define a local coordinate system that you can use when specifying loads and constraints.

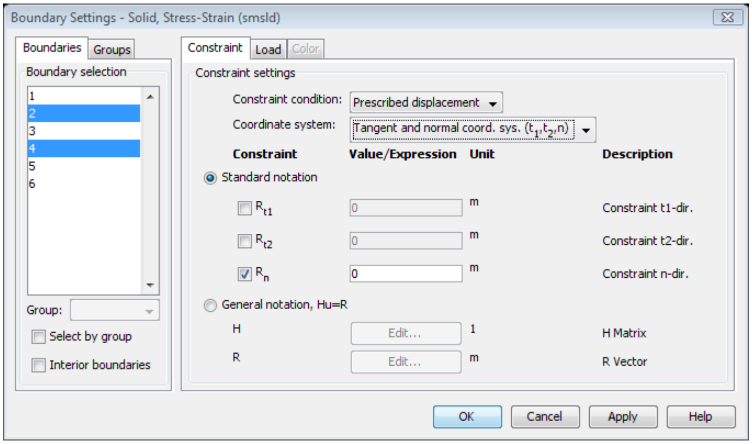
In 2D, the local geometrical coordinate system (t, n) represents the directions tangential and normal to the boundary. For interior boundaries and free edges this coordinate system is right-oriented. For exterior boundaries the normal is always directed out from the domain.

In 3D, the local geometrical coordinate system (t_1, t_2, n) represents two tangential directions and one normal direction. t_1 and t_2 depend on the parametrization of the geometry. For interior boundaries and free faces this coordinate system is right-oriented but not always orthogonal. For exterior boundaries the normal is always directed out from the domain. Common applications for this coordinate system include specifying pressure or normal displacement on a surface.

Note: t_1 and t_2 depend on how the geometry was created and are usually perpendicular to each other.

Read more about this topic in “Geometric Variables” on page 165 in the *COMSOL Multiphysics User’s Guide*.

The **Constraint** page in the **Boundary Settings** dialog box for the Solid, Stress-Strain application mode shows how local coordinate systems work.



Application-Mode Specific Coordinate Systems

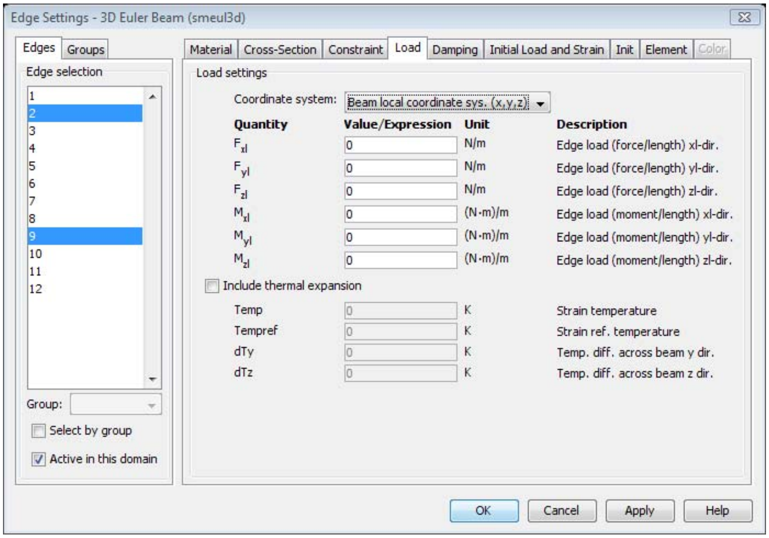
The 3D Euler Beam and Shell application modes include a local coordinate system.

The 3D Euler Beam application mode defines the orientation of the cross-section coordinate system needed to specify the orientation of the beam. Details about the 3D Euler beam local coordinate system is available in “Cross Section” on page 263.

The Shell application mode defines a local coordinate system on the face needed to define postprocessing variables such as internal moments, normal forces, and shear forces. Details about the shell local coordinate system are found in “Postprocessing” on page 316 of this manual.

You can also use these coordinate systems to define loads and constraints.

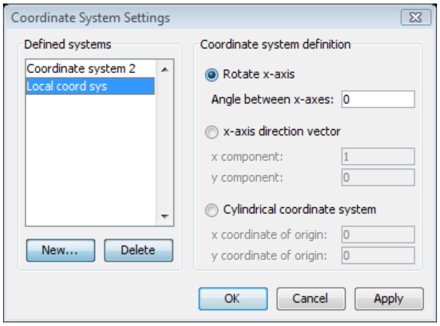
The **Load** page in the **Edge** settings dialog box for the 3D Euler Beam application mode shows how local application-specific coordinate systems is used.



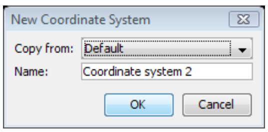
User-Defined Coordinate Systems

User-defined coordinate systems can be applied at all domain levels in all application modes. For the continuum application modes, they can define orthotropic and anisotropic material properties in a coordinate system other than the global Cartesian system.

Create a user-defined coordinate system by choosing **Options>Coordinate Systems**, thereby opening the **Coordinate Systems Settings** dialog box. Depending on the active geometry, the software creates a 2D or 3D coordinate system.



The **New** button opens the **New Coordinate System** dialog box.

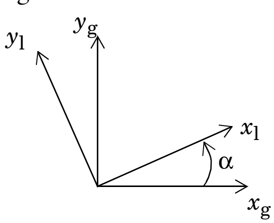


In the **Copy from** list you select from which existing coordinate system you want to copy the coordinate-system settings.

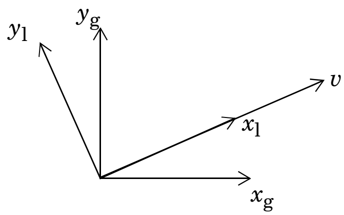
In the **Name** edit field you enter the name of the coordinate system, and it is the name that appears in all coordinate-system lists.

The software creates a coordinate system in one of three ways, which you control with option buttons:

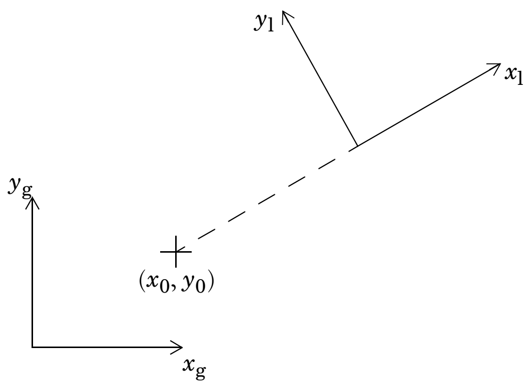
- **Rotate x-axis**: The local x_l-axis direction is specified by an angle (α) between the global and local x-axes.



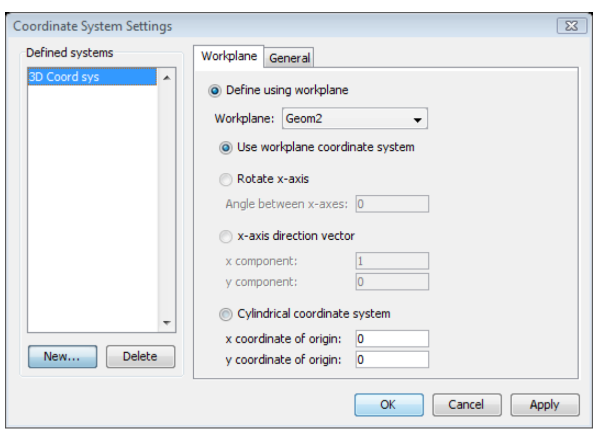
- **x-axis direction vector**: Specify the local x_l-axis direction by a direction vector v.



- **Cylindrical coordinate system**: A local cylindrical coordinate system (x_l, y_l) with origin at (x_0, y_0) is specified.



3D GEOMETRY



The **New** button works in the same way as for the 2D geometry case.

The software defines the coordinate system in one of two ways, which you control with the **Define using work plane** and **Define using global coordinates** option buttons.

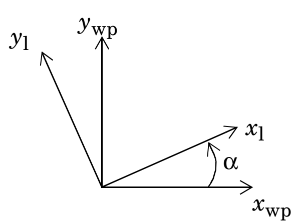
Define Using Work Plane

Define using work plane is enabled when a least one work plane/2D geometry exists.

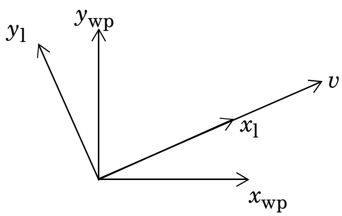
Select the work plane on which to base the local coordinate system from the **Work plane** list.

Four options are available, which you control with option buttons:

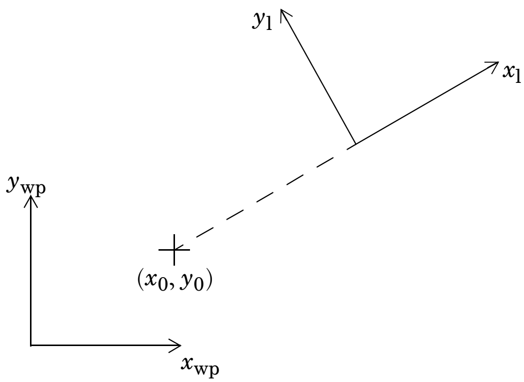
- **Use work plane coordinate system**: The local coordinate system is the same as the work plane. You control the definition of the work plane by going to the **Draw** menu and opening the **Work-Plane Settings** dialog box. Get details about the creation of work planes in "Creating and Using 2D Work Planes" on page 59 in the *COMSOL Multiphysics User's Guide*.
- **Rotate x-axis**:The local x_l-axis direction is specified by an angle (α) between the work planes x_{wp}-axis and the local x_l-axis.



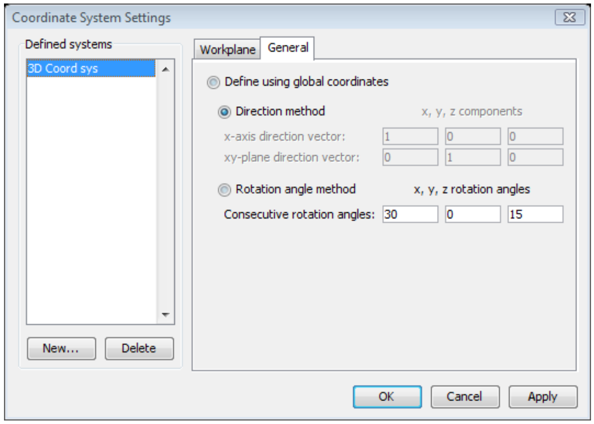
- **x-axis direction vector**: The local x_l-axis direction is specified by a direction vector v.



- **Cylindrical coordinate system**: A local cylindrical coordinate system (x_l, y_l) with origin at (x_0, y_0) in the work plane coordinates is specified.

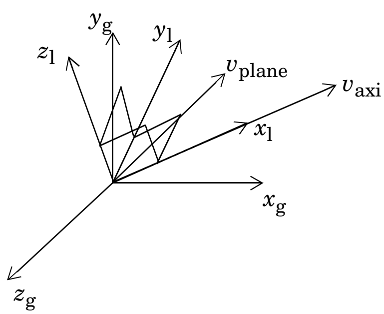


Define Using Global Coordinates

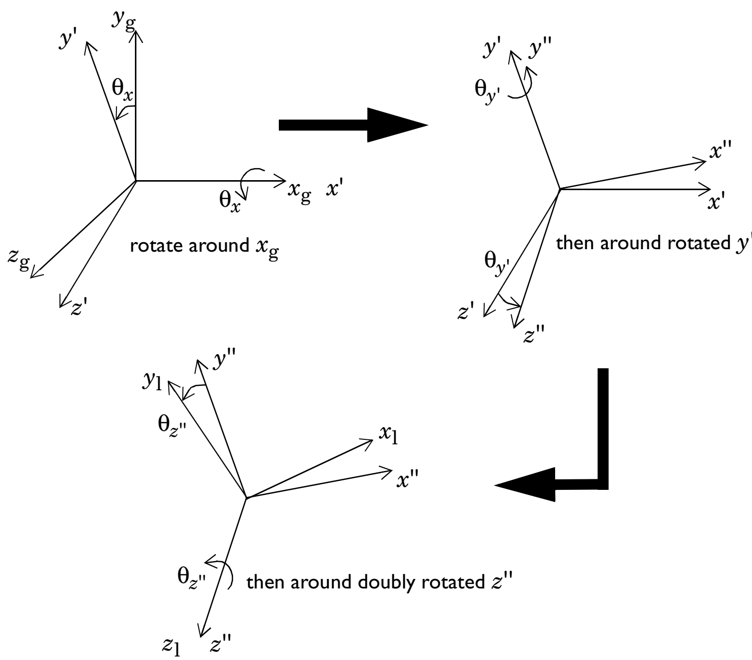


There are two different options available, which you control with option buttons:

- **Direction method**: The local x_l-axis direction is specified by a direction vector v_{axi}. The local $x_l y_l$-plane is specified using a direction vector v_{plane} which is a vector lying in the local $x_l y_l$-plane.



- **Rotation angle method**: The local coordinate system (x_l, y_l, z_l) is specified using three consecutive rotation angles θ_x, $\theta_{y'}$, and $\theta_{z''}$.



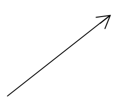
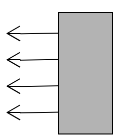
Symbols for Loads and Constraints

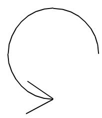
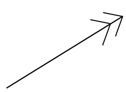
To make it easier to specify a model, you can choose to display load and constraint symbols on a geometry. This is done automatically, but in some situations it might take too long, so the software provides an option to deactivate the automatic update of symbols. This option appears on the **Visualization** page in the **Preferences** dialog box. To read more about that dialog box see the section “Saving Preferences for Labels, Rendering, and Highlighting” on page 119 in the *COMSOL Multiphysics User’s Guide*. In the **Preferences** dialog box you also have the option to select whether to plot the symbols from the current domain type or all domain types. A manual update of symbols is possible from the **Options** menu by selecting **Update Symbols** or by clicking the **Update Symbols** button on the Visualization/Selection toolbar. Scaling the size of the symbols is possible in the **Visualization/Selection** dialog box; see “Scaling of Load and Constraint Symbols” on page 119 in the *COMSOL Multiphysics User’s Guide*.

Load Symbols

You can plot load symbols on points, boundaries, edges, and subdomains. The loads are normalized with respect to the maximum value within a domain type.

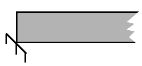
The following table lists all load symbols together with the application modes where they appear.

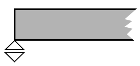
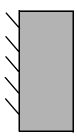
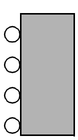
LOAD SYMBOL	DESCRIPTION	APPLICATION MODES
	Force at a point or in a subdomain.	Plane Stress, Piezo Plane Stress, Plane Strain, Piezo Plane Strain, Axial Symmetry, Stress-Strain, Piezo Axial Symmetry, 2D Euler Beam
	Boundary force.	Plane Stress, Piezo Plane Stress, Plane Strain, Piezo Plane Strain, Axial Symmetry, Stress-Strain, Piezo Axial Symmetry, 2D Euler Beam

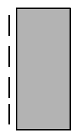
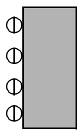
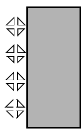
LOAD SYMBOL	DESCRIPTION	APPLICATION MODES
	Transversal force in the z direction at point or in a subdomain. Positive (left) and negative (right) z direction respectively.	Mindlin Plate
	Point bending moment about the z-axis.	2D Euler Beam
	Edge bending moment about the z-axis.	2D Euler Beam
	Bending moment about the axis indicated by the direction of the arrow.	Mindlin Plate
	Force in the direction indicated by the direction of the arrow.	Solid, Stress-Strain, Piezo Solid, Shell, 3D Euler Beam
	Moment about the axis indicated by the direction of the arrow.	Shell, 3D Euler Beam
Fl	Force and/or moment defined in the local coordinate system.	3D Euler Beam, local coordinate system.

Constraint Symbols

Constraint symbols are available on points, boundaries, edges, and subdomains. The following table lists all constraint symbols together with the application modes where they appear.

CONSTRAINTS SYMBOL	DESCRIPTION	APPLICATION MODE
	Displacement constrained in the direction indicated by the roller.	Plane Stress, Piezo Plane Stress, Plane Strain, Piezo Plane Strain, Axial Symmetry, Stress-Strain, Piezo Axial Symmetry, 2D Euler Beam
	Displacement constrained in the x and y directions.	2D Euler Beam
	All degrees of freedom constrained.	Plane Stress, Piezo Plane Stress, Plane Strain, Piezo Plane Strain, Axial Symmetry, Stress-Strain, Piezo Axial Symmetry, 2D Euler Beam
	Rotation constrained.	2D Euler Beam
	Rotation constrained. Displacement constrained in the direction indicated by the roller.	2D Euler Beam

CONSTRAINTS SYMBOL	DESCRIPTION	APPLICATION MODE
	Displacement constrained in the z direction.	Mindlin Plate
	Displacement constrained in the z direction. Rotation constrained but allowed about the axis indicated by the line segments.	Mindlin Plate
	Rotation constrained about the axis indicated by the space between the triangles.	Mindlin Plate
	Rotations about all axes constrained.	Mindlin Plate
	Clamped edge, all degrees of freedom constrained.	Plane Stress, Piezo Plane Stress, Plane Strain, Piezo Plane Strain, Axial Symmetry, Stress-Strain, Piezo Axial Symmetry, 2D Euler Beam
	Displacement constrained in the z direction.	Mindlin Plate

CONSTRAINTS SYMBOL	DESCRIPTION	APPLICATION MODE
	Rotation constrained but allowed about the axis indicated by the line segments.	Mindlin Plate
	Displacement constrained in the z direction. Rotation constrained but allowed about the axis indicated by the line segments.	Mindlin Plate
	Rotations about all axes constrained.	Mindlin Plate
	Displacements constrained in the directions indicated by the arrows.	3D Euler Beam, Shell, Solid, Stress-Strain, Piezo Solid
	Rotations constrained about axis directions indicated by the arrows.	3D Euler Beam Shell
CI	Displacements and/or rotations constrained in the local coordinate system.	3D Euler Beam, local coordinate system.

7

Continuum Application Modes

Continuum in this context means that no simplifications are available and that you solve for the displacements without involving rotations.

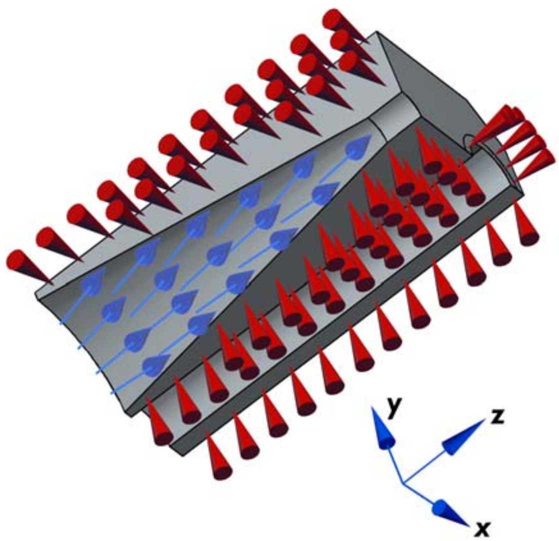
The following application modes in the Structural Mechanics Module are of the continuum type:

- Solid, Stress-strain
- Plane Stress
- Plane Strain
- Axial Symmetry, Stress-Strain

Continuum application modes are formulated on planes in 2D and volumes in 3D. In the continuum application modes you can use Lagrange elements of arbitrary order.

Solid, Stress-Strain

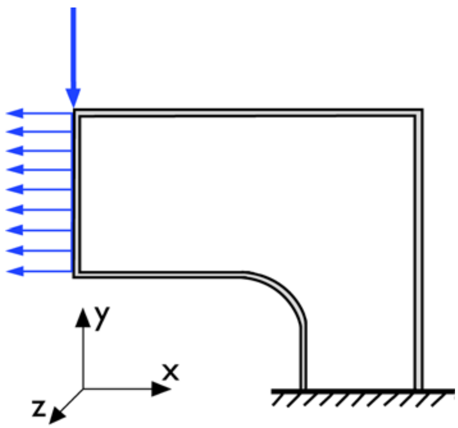
The degrees of freedom (dependent variables) in this application mode are the global displacements u, v, and w in the global x, y, and z directions, respectively, and the pressure p (only used if mixed formulation is selected).



Loads and constraints applied to a 3D solid using the Solid, Stress-Strain application mode.

Plane Stress

Use the Plane Stress application mode in the Structural Mechanics Module to analyze thin in-plane loaded plates. It solves for the global displacements (u, v) in the x and y directions, the pressure p (only used for mixed formulation), and the displacement derivative in the perpendicular direction (only used for hyperelastic material). For a state of plane stress, this mode assumes the σ_z, τ_{yz}, and τ_{xz} components of the stress tensor are zero.

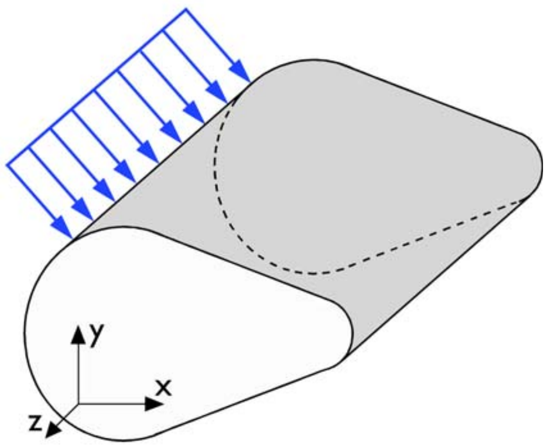


The Plane Stress application mode models plates where the loads are only in the plane; it does not include any out-of-plane stress components.

This application mode allows loads in the x and y directions, and it assumes that these are constant throughout the material's thickness, which however can vary with x and y. The plane stress condition prevails in a thin flat plate in the xy-plane loaded only in its own plane and without any z direction restraint.

Plane Strain

The Plane Strain application mode in the Structural Mechanics Module solves for the global displacements (u, v) in the x and y directions and the pressure p (only if mixed formulation is used). The assumption that defines a state of plane strain is that the ε_z, ε_{yz}, and ε_{xz} components of the strain tensor are zero.



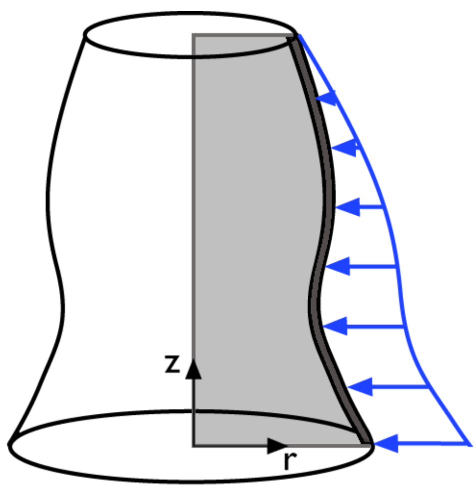
A geometry suitable for plane strain analysis.

Loads in the x and y directions are allowed. The loads are assumed to be constant throughout the thickness of the material, but the thickness can vary with x and y. The plane strain condition prevails in geometries, whose extent is large in the z direction compared to in the x and y directions, or when the z displacement is in some way restricted. One example is a long tunnel along the z-axis where it is sufficient to study a unit-depth slice in the xy-plane.

Axial Symmetry, Stress-Strain

The Axial Symmetry, Stress-Strain application mode uses cylindrical coordinates r, φ (`phi`), and z. It solves equations for the global displacement (uor, w) in the r and z directions and the pressure p (only used for mixed formulation). The dependent variable, $uor = u/r$, is introduced to avoid division by r, which causes problems on the axis where $r = 0$. This application mode assumes that the displacement v in the φ direction together with the $\tau_{r\varphi}$, $\tau_{\varphi z}$, $\gamma_{r\varphi}$, and $\gamma_{\varphi z}$ components of the stresses and strains are zero. Loads are independent of φ, and this application mode allows loads only in the r and z directions.

You can view the domain where the application mode solves the equations as the intersection between the original axially symmetric 3D solid and the half plane $\varphi = 0$, $r \geq 0$. Therefore you draw the geometry only in the half plane $r \geq 0$ and recover the original 3D solid by rotating the 2D geometry about the z-axis.



Rotating a 2D geometry to recover a 3D solid.

Note: $r = 0$ is the symmetry axis. In the Axisymmetry, Stress-Strain application mode $x \to r$ and $y \to z$.

Theory Background

Strain-Displacement Relationship

The strain consists of thermal (ε_{th}), elastic (ε_{el}), and initial (ε_0) contributions so that

$$\varepsilon = \varepsilon_{el} + \varepsilon_{th} + \varepsilon_0$$

The strain conditions at a point are completely defined by the deformation components—u, v, and w in 3D—and their derivatives. The precise relation between strain and deformation depends on the relative magnitude of the displacement.

SMALL DISPLACEMENTS

Under the assumption of small displacements, the normal strain components and the shear strain components are related to the deformation as follows:

$$\begin{aligned}
\varepsilon_x &= \frac{\partial u}{\partial x} & \varepsilon_{xy} &= \frac{\gamma_{xy}}{2} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \\
\varepsilon_y &= \frac{\partial v}{\partial y} & \varepsilon_{yz} &= \frac{\gamma_{yz}}{2} = \frac{1}{2}\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right) \\
\varepsilon_z &= \frac{\partial w}{\partial z} & \varepsilon_{xz} &= \frac{\gamma_{xz}}{2} = \frac{1}{2}\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right).
\end{aligned} \tag{7-1}$$

To express the shear strain, use either the tensor form, ε_{xy}, ε_{yz}, ε_{xz}, or the engineering form, γ_{xy}, γ_{yz}, γ_{xz}.

The symmetric strain tensor ε consists of both normal and shear strain components:

$$\varepsilon = \begin{bmatrix} \varepsilon_x & \varepsilon_{xy} & \varepsilon_{xz} \\ \varepsilon_{xy} & \varepsilon_y & \varepsilon_{yz} \\ \varepsilon_{xz} & \varepsilon_{yz} & \varepsilon_z \end{bmatrix}$$

The strain-displacement relationships for the axial symmetry case for small displacements are

$$\varepsilon_r = \frac{\partial u}{\partial r}, \qquad \varepsilon_\varphi = \frac{u}{r}, \qquad \varepsilon_z = \frac{\partial w}{\partial z}, \text{ and } \qquad \gamma_{rz} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial r}$$

LARGE DISPLACEMENTS

For large-displacement analysis, the deformation is not small and you calculate the strains without this restriction. The resulting strains are known as Green or *Green-Lagrange strains*, and large displacement is sometimes referred to as *geometric nonlinearity* or *nonlinear geometry*.

Green strains are defined with reference to an undeformed geometry. Hence, they represent a Lagrangian description.

In a small-strain, large rotational analysis, the Green strain corresponds to the engineering strain in directions that follow the deformed body. The Green strain is a natural choice when formulating a problem in the undeformed state.

The Green strain components, ε_{ij}, are

$$\varepsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} + \frac{\partial u_k}{\partial x_i} \cdot \frac{\partial u_k}{\partial x_j}\right) \qquad (7\text{-}2)$$

ANALYSIS OF DEFORMATION AND THE DEFORMATION GRADIENT

As a start, consider a certain physical particle, initially located at the coordinate $\mathbf{X}$. During deformation, this particle follows a path

$$\mathbf{x} = \mathbf{x}(\mathbf{X}, t)$$

For simplicity, assume that undeformed and deformed positions are measured in the same coordinate system. Using the displacement $\mathbf{u}$, it is then possible to write

$$\mathbf{x} = \mathbf{X} + \mathbf{u}$$

When studying how an infinitesimal line element $d\mathbf{X}$ is mapped to the corresponding deformed line element $d\mathbf{x}$, the *deformation gradient* F defined by

$$d\mathbf{x} = \frac{\partial \mathbf{x}}{\partial \mathbf{X}} d\mathbf{X} = F d\mathbf{X}$$

is used.

The deformation gradient contains the complete information about the local straining and rotation of the material. It is a positive definite matrix, as long as material cannot be annihilated. The ratio between current and original volume (or mass density) is

$$\frac{dV}{dV_0} = \frac{\rho_0}{\rho} = \det(F) = J$$

As a consequence, a deformation state where $J = 1$ is said to be incompressible. From the deformation gradient, it is possible to define the right Cauchy-Green tensor as

$$C = F^T F$$

The most commonly used definition of strain is the *engineering strain* ε; see Equation 7-1.

As can be shown by simple insertion, a finite rigid body rotation will cause nonzero values of the engineering strain. This is not in correspondence with the intuitive concept of strain, and it is certainly not useful in a constitutive law. There are several alternative strain definitions in use that do have the desired properties. The Green strain, ε, is defined as

$$\varepsilon = \frac{1}{2}(C - I) = \frac{1}{2}(F^T F - I)$$

Using the displacements, the Green strain can be also written as shown in Equation 7-2.

The deformation gradient and its inverse are available as variables and can be used, for instance, to model follower loads; see the Hyperelastic Seal model on page 467 in the *Structural Mechanics Model Library* for an example.

Stress-Strain Relationship

The symmetric stress tensor σ describes stress in a material:

$$\sigma = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix} \qquad \tau_{xy} = \tau_{yx} \qquad \tau_{xz} = \tau_{zx} \qquad \tau_{yz} = \tau_{zy}$$

This tensor consists of three normal stresses (σ_x, σ_y, σ_z) and six (or, if symmetry is used, three) shear stresses (τ_{xy}, τ_{yz}, τ_{xz}).

For large deformations and hyperelastic material models there are more than one stress measure:

- Cauchy stress σ (the components are denoted `sx`, … in COMSOL Multiphysics) defined as force/deformed area in fixed directions not following the body. Symmetric tensor.

- First Piola-Kirchhoff stress P (the components are denoted `Px`, … in COMSOL Multiphysics). This is an unsymmetric tensor used only for hyperelastic material models.
- Second Piola-Kirchhoff stress S (the components are denoted `Sx`, … in COMSOL Multiphysics). This is a symmetric tensor, for small strains same as Cauchy stress tensor but in directions following the body.

The stresses relate to each other as

$$S = F^{-1}P$$

$$\sigma = J^{-1}PF^T = J^{-1}FSF^T$$

LINEAR ELASTIC MATERIAL

The stress-strain relationship—or the *constitutive equation*—for linear conditions including initial stress and strain and thermal effects reads:

$$\sigma = D\varepsilon_{\mathrm{el}} + \sigma_0 = D(\varepsilon - \varepsilon_{\mathrm{th}} - \varepsilon_0) + \sigma_0$$

where D is the 6-by-6 elasticity matrix and the stress and the strain are both given in column vector form:

$$\sigma = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{bmatrix} \qquad \varepsilon = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix}$$

Note: In the following descriptions σ and ε denote either the stress and strain vectors or the corresponding tensors depending on the circumstances.

The elasticity matrix D—or the more basic flexibility (or compliance) matrix, the inverse of D—is defined differently for isotropic, orthotropic, and anisotropic materials. For an isotropic material, the flexibility matrix looks like

$$
D^{-1} = \frac{1}{E}\begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2(1+\nu) & 0 & 0 \\ 0 & 0 & 0 & 0 & 2(1+\nu) & 0 \\ 0 & 0 & 0 & 0 & 0 & 2(1+\nu) \end{bmatrix}
$$

where E is the modulus of elasticity or *Young's modulus*, and ν is *Poisson's ratio*, which defines the contraction in the perpendicular direction. Inverting D^{-1} results in the following elasticity matrix:

$$
D = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix}
$$

For an orthotropic material, the D^{-1} matrix takes the form

$$
D^{-1} = \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{yx}}{E_y} & -\frac{\nu_{zx}}{E_z} & 0 & 0 & 0 \\ -\frac{\nu_{xy}}{E_x} & \frac{1}{E_y} & -\frac{\nu_{zy}}{E_z} & 0 & 0 & 0 \\ -\frac{\nu_{xz}}{E_x} & -\frac{\nu_{yz}}{E_y} & \frac{1}{E_z} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{xy}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{yz}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{xz}} \end{bmatrix}
$$

where you supply the values of E_x, E_y, E_z, ν_{xy}, ν_{yz}, ν_{xz}, G_{xy}, G_{yz}, and G_{xz} in designated edit fields in the user interface. The software deduces the remaining components—ν_{yx}, ν_{zx}, and ν_{zy}—by using the fact that the matrices D and D^{-1} are symmetric.

Note: The definitions of the components ν_{ij} can vary depending on the application field. When specifying the material properties, make sure you use the definitions just given. If necessary, transform your material data so that it conforms with the above conventions before entering it in the Structural Mechanics Module's user interface.

Inverting D^{-1} using only the E_x, E_y, E_z, ν_{xy},ν_{yz}, ν_{xz}, G_{xy}, G_{yz}, and G_{xz} coefficients results in the symmetric D matrix

$$D = \begin{bmatrix} D_{11} & D_{12} & D_{13} & 0 & 0 & 0 \\ D_{12} & D_{22} & D_{23} & 0 & 0 & 0 \\ D_{13} & D_{23} & D_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & D_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & D_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & D_{66} \end{bmatrix}$$

where the components are as follows:

$$D_{11} = \frac{E_x^2(E_z\nu_{yz}^2 - E_y)}{D_{\mathrm{denom}}}, \qquad D_{12} = -\frac{E_xE_y(E_z\nu_{yz}\nu_{xz} + E_y\nu_{xy})}{D_{\mathrm{denom}}},$$

$$D_{13} = -\frac{E_xE_yE_y(\nu_{xy}\nu_{yz} + \nu_{xz})}{D_{\mathrm{denom}}}, \qquad D_{22} = \frac{E_y^2(E_z\nu_{xz}^2 - E_x)}{D_{\mathrm{denom}}},$$

$$D_{23} = -\frac{E_yE_z(E_y\nu_{xy}\nu_{xz} + E_x\nu_{yz})}{D_{\mathrm{denom}}}, \qquad D_{33} = \frac{E_yE_z(E_y\nu_{xy}^2 - E_x)}{D_{\mathrm{denom}}},$$

$$D_{44} = G_{xy}, \quad D_{55} = G_{yz}, \text{ and } D_{66} = G_{xz}$$

where

$$D_{\mathrm{denom}} = E_yE_z\nu_{xz}^2 - E_xE_y + 2\nu_{xy}\nu_{yz}\nu_{xz}E_yE_z + E_xE_z\nu_{yz}^2 + E_y^2\nu_{xy}^2$$

For an anisotropic material, you provide the symmetric D matrix explicitly.

Mixed Formulation

Mixed formulations are used for nearly incompressible materials. The solution is to add the negative mean stress as a new dependent variable, p (pressure).

$$p = -\left(\frac{\sigma_x + \sigma_y + \sigma_z}{3}\right)$$

The stress-strain relation for linear elastic material for 3D is

$$\sigma = D\varepsilon_{\mathrm{el}} + \sigma_0 = D(\varepsilon - \varepsilon_{\mathrm{th}} - \varepsilon_0) + \sigma_0$$

The stress σ is separated in a deviatoric part, σ_{d}, and a mean part, $-p$:

$$\sigma = \sigma_{\mathrm{d}} - mp$$

where

$$\sigma_{\mathrm{d}} = D_{\mathrm{d}}(\varepsilon - \varepsilon_{\mathrm{th}} - \varepsilon_0) + \sigma_{0\mathrm{d}}$$

$$\sigma_0 = \sigma_{0\mathrm{d}} - p_0$$

$$p_0 = -\left(\frac{\sigma_{0x} + \sigma_{0y} + \sigma_{0z}}{3}\right)$$

and m is a six-dimensional column vector. Inserting the stress-strain relation for isotropic materials results in the following expressions for the deviatoric stress and the vector m:

$$\sigma_{\mathrm{d}} = D_{\mathrm{d}}\left[\begin{bmatrix}\varepsilon_x\\ \varepsilon_y\\ \varepsilon_z\\ \varepsilon_{xy}\\ \varepsilon_{yz}\\ \varepsilon_{xz}\end{bmatrix} - \begin{bmatrix}\varepsilon_{x0}\\ \varepsilon_{y0}\\ \varepsilon_{z0}\\ \varepsilon_{xy0}\\ \varepsilon_{yz0}\\ \varepsilon_{xz0}\end{bmatrix} - \alpha(T - T_{\mathrm{ref}})m\right] + (\sigma_0 + p_0)$$

where

$$D_{\mathrm{d}} = G\begin{bmatrix} \frac{4}{3} & -\frac{2}{3} & -\frac{2}{3} & 0 & 0 & 0 \\ -\frac{2}{3} & \frac{4}{3} & -\frac{2}{3} & 0 & 0 & 0 \\ -\frac{2}{3} & -\frac{2}{3} & \frac{4}{3} & 0 & 0 & 0 \\ 0 & 0 & 0 & 2 & 0 & 0 \\ 0 & 0 & 0 & 0 & 2 & 0 \\ 0 & 0 & 0 & 0 & 0 & 2 \end{bmatrix}$$

$$m = \begin{bmatrix} 1 \\ 1 \\ 1 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

The equation for the pressure becomes

$$\frac{p}{K} + m^T(\varepsilon - \varepsilon_{\mathrm{th}} - \varepsilon_0) - \frac{p_0}{K} = 0$$

$$K = \frac{E}{3(1-2\nu)}$$

where G is the shear modulus and K is the bulk modulus.

For orthotropic and anisotropic materials some scaling is performed to get a system of equations that produces symmetric matrices. The equations for the stress and the pressure become

$$\sigma = \sigma_{\mathrm{d}} - np$$

$$\sigma_{\mathrm{d}} = D_{\mathrm{d}}(\varepsilon - \varepsilon_{\mathrm{th}} - \varepsilon_0) + \sigma_{0\mathrm{d}}$$

$$\sigma_0 = \sigma_{0\mathrm{d}} - p_0$$

$$p_0 = -\left(\frac{\sigma_{0x} + \sigma_{0y} + \sigma_{0z}}{3}\right)$$

$$\frac{9p}{D_{\mathrm{sum}}} + m^T(\varepsilon - \varepsilon_{\mathrm{th}} - \varepsilon_0) - \frac{9p_0}{D_{\mathrm{sum}}} = 0$$

where, n, D_{d}, and m are defined as

$$m_i = \frac{3d_i}{D_{\mathrm{sum}}}$$

$$n_i = m_i \qquad i = 1, \ldots, 3$$
$$n_i = 0.5m_i \qquad i = 4, \ldots, 6$$

$$D_{\mathrm{sum}} = \sum_{\substack{i = 1, \ldots, 3 \\ j = 1, \ldots, 6}} D_{ij}$$

$$d_i = D_{1i} + D_{2i} + D_{3i}$$

$$D_{\mathrm{d}ij} = D_{ij} - \frac{d_i d_j}{D_{\mathrm{sum}}} \qquad i = 1, \ldots, 3 \qquad j = 1, \ldots, 6$$
$$D_{\mathrm{d}ij} = D_{ij} - 0.5\frac{d_i d_j}{D_{\mathrm{sum}}} \qquad i = 4, \ldots, 6 \qquad j = 1, \ldots, 6$$

This produces symmetric matrices.

Note: The mixed formulation creates indefinite matrices, which affects the selection of iterative solvers. See "Selecting Iterative Solvers" on page 127 for more information.

The mixed formulation are useful not only for linear elastic material but also for elasto-plastic and hyperelastic materials.

In the mixed formulation the pressure is interpolated using a polynomial of one order less than the one used for the displacement variables.

If loss factor damping is used with frequency response analysis, the loss information appears also in the pressure equation. The equation with loss factor damping for isotropic material is

$$\frac{p}{(1 + j\eta)K} + m^T\varepsilon - \frac{m^T(\varepsilon_{\mathrm{th}} + \varepsilon_0)}{(1 + j\eta)} - \frac{p_0}{(1 + j\eta)K} = 0$$
$$K = \frac{E}{3(1 - 2\nu)}$$

and the equation for orthotropic and anisotropic materials is

$$\frac{9p}{(1+j\eta)D_{\mathrm{sum}}}+m^T\varepsilon-\frac{m^T(\varepsilon_{\mathrm{th}}+\varepsilon_0)}{(1+j\eta)}-\frac{9p_0}{(1+j\eta)D_{\mathrm{sum}}} = 0$$

where η is the loss factor.

ELASTO-PLASTIC MATERIALS

In an elasto-plastic material the stress-strain relationship is

$$\sigma = D\varepsilon_{\mathrm{el}}+\sigma_0 = D(\varepsilon-\varepsilon_{\mathrm{p}}-\varepsilon_{\mathrm{th}}-\varepsilon_0)+\sigma_0$$

where ε_{p} is the *plastic strain* vector.

The variable ε_{p} and a vector κ of *state parameters* describe the state of a plastic deformation. To describe the evolution of these variables, use the rate equations

$$\dot{\varepsilon}_{\mathrm{p}} = \lambda H(\varepsilon_{\mathrm{p}}, \kappa, v), \qquad \dot{\kappa} = \lambda G(\varepsilon_{\mathrm{p}}, \kappa, v)$$

where v is a vector whose variables form the solution vector (with parameters such as displacements and temperature) and λ is the *plastic multiplier*. The dot stands for differentiation with respect to pseudo-time or time. The plastic multiplier is determined by the *complementarity conditions*

$$F(\varepsilon_{\mathrm{p}}, \kappa, v)\le 0, \qquad \lambda\ge 0, \qquad F(\varepsilon_{\mathrm{p}}, \kappa, v)\lambda = 0$$

where F is the *yield function*. The functions F, G, and H often take a simpler form when expressed in terms of the *generalized stress*, Σ,

$$\Sigma = \begin{bmatrix}\sigma\\ \chi\end{bmatrix} = \begin{bmatrix}f_1(\varepsilon-\varepsilon_{\mathrm{p}}, v)\\ f_2(\varepsilon_{\mathrm{p}}, \kappa, v)\end{bmatrix}$$

where σ is the vector of stress components, and χ is the vector of *conjugate forces*. The function f_1 is often a linear function (matrix D). For *associated plasticity*, which is the rule,

$$H(\sigma,\chi) = \frac{\partial}{\partial\sigma}F(\sigma,\chi), \qquad G(\sigma,\chi) = -\frac{\partial}{\partial\chi}F(\sigma,\chi)$$

For *non-associated plasticity*, which is very uncommon,

$$H(\sigma,\chi) = \frac{\partial}{\partial\sigma}Q(\sigma,\chi), \qquad G(\sigma,\chi) = -\frac{\partial}{\partial\chi}Q(\sigma,\chi)$$

where Q is a *plastic potential.*

Now consider some special cases where the *effective stress function*, ϕ , is often the *von Mises stress*.

Perfect (or Ideal) Plasticity

$$\kappa = \text{empty}, \qquad F = \phi(\sigma) - Y_0 \qquad H = \frac{\partial F}{\partial \sigma}$$

where Y_0 is the *yield stress*.

Isotropic Hardening

$$\kappa = \varepsilon_{\mathrm{pe}}, \qquad \chi = Y = f_2(\varepsilon_{\mathrm{pe}}), \qquad F(\sigma, Y) = \phi(\sigma) - Y \qquad G = -\frac{\partial F}{\partial \sigma} = 1$$

where $\varepsilon_{\mathrm{pe}}$ is the *effective plastic strain*, and Y is the yield stress. The function f_2 is often nonlinear.

Kinematic Hardening

$$\kappa = \varepsilon_{\mathrm{p}}, \qquad \chi = \sigma_{\mathrm{shift}} = f_3(\varepsilon_p), \qquad F = \phi(\sigma - \sigma_{\mathrm{shift}}) - Y_0, \qquad G = -\frac{\partial F}{\partial \sigma_{\mathrm{shift}}}$$

where f_3 often is a linear function.

In cases with kinematic hardening, the plastic strain is a subset of the state parameters. Then you only need the rate equation

$$\dot{\kappa} = \lambda G(\kappa, v)$$

and the complementarity conditions

$$F(\kappa, v) \le 0, \qquad \lambda \ge 0, \qquad F(\kappa, v)\lambda = 0$$

You can thus write the generalized stress, Σ, as

$$\Sigma = \begin{bmatrix} \sigma \\ \chi \end{bmatrix} = f(\kappa, v)$$

These formulas also hold for cases without kinematic hardening if you enlarge the vector of state parameters to include the plastic strain. From now on this discussion assumes this definition of κ, leading to the following special cases:

Perfect (or Ideal) Plasticity

$$\kappa = \varepsilon_{\mathrm{p}}, \qquad F = \phi(\sigma) - Y_0, \qquad G = \frac{\partial F}{\partial \sigma}$$

Isotropic Hardening

$$\kappa = \begin{bmatrix} \varepsilon_{\mathrm{p}} \\ \varepsilon_{\mathrm{pe}} \end{bmatrix}, \qquad \chi = Y = f_2(\varepsilon_{\mathrm{pe}}), \qquad F = \phi(\sigma) - Y, \qquad G = \begin{bmatrix} \dfrac{\partial F}{\partial \sigma} \\ -\dfrac{\partial F}{\partial Y} \end{bmatrix}$$

where $\varepsilon_{\mathrm{pe}}$ is the effective plastic strain, and Y is the yield stress. The function f_2 is often nonlinear.

Kinematic Hardening

$$\kappa = \varepsilon_{\mathrm{p}}, \qquad \chi = \sigma_{\mathrm{shift}} = f_3(\varepsilon_p), \qquad F = \phi(\sigma - \sigma_{\mathrm{shift}}) - Y_0, \qquad G = \frac{\partial F}{\partial \sigma}$$

where f_3 often is a linear function.

Postprocessing

The effective plastic strain is calculated from the equation

$$\varepsilon_{\mathrm{pe}} = \int_0^t \dot{\varepsilon}_{\mathrm{pe}} dt$$

$$\dot{\varepsilon}_{\mathrm{pe}} = \frac{\sqrt{2}}{3}\sqrt{(\dot{\varepsilon}_{\mathrm{p}x} - \dot{\varepsilon}_{\mathrm{p}y})^2 + (\dot{\varepsilon}_{\mathrm{p}y} - \dot{\varepsilon}_{\mathrm{p}z})^2 + (\dot{\varepsilon}_{\mathrm{p}x} - \dot{\varepsilon}_{\mathrm{p}z})^2 + 6\dot{\varepsilon}_{\mathrm{p}xy}^{\;2} + 6\dot{\varepsilon}_{\mathrm{p}yz}^{\;2} + 6\dot{\varepsilon}_{\mathrm{p}xz}^{\;2}}$$

The plastic strain can be calculated in the node points like any other variable but this may cause problems because evaluating the plastic strain involves solving an equation system.

For postprocessing purposes, Gauss-point variables are an alternative to the stresses, plastic strains, and effective plastic strain discussed above. Gauss-point variables are normally better because they are the values that were calculated during the solution process. The Gauss-point variables have the suffix `Gp` appended to their names, for example, `sxGp` instead of `sx`.

The elasto-plastic material model requires that you use a solver that can follow the load history, that is, you need to use the nonlinear parametric or transient solver. You cannot use the nonlinear static solver together with an elasto-plastic material model.

HYPERELASTIC MATERIALS

A *hyperelastic material* is defined from its strain energy function, W_s, which is a function of the strain state. The stress in such a material is computed from the strain energy function W_s, In the following, assume that the First Piola-Kirchhoff stresses $\boldsymbol{P}$ and the displacement gradient $\nabla\mathbf{u}$ are used, so that

$$P = \frac{\partial W_s}{\partial \nabla\mathbf{u}} \tag{7-3}$$

For an isotropic material, W_s can only be a function of the strain invariants. In a total Lagrangian formulation it is convenient to use the right Cauchy-Green tensor $\boldsymbol{C} = \boldsymbol{F}^T\boldsymbol{F}$ for the representation of the strain. The invariants are:

$$I_1 = \mathrm{trace}(C) = C_{11} + C_{22} + C_{33}$$

$$I_2 = \frac{1}{2}(I_1^2 - \mathrm{trace}(C^2))$$

$$I_3 = \det(C) = J^2$$

where $\boldsymbol{J} \equiv \det(\boldsymbol{F})$. Due to the incompressibility, it is often a good idea to work with modified invariants, where the two first invariants have no dependency on the volume change. Such invariants can be defined as

$$\bar{I}_1 = I_1 J^{-\frac{2}{3}}$$

$$\bar{I}_2 = I_2 J^{-\frac{4}{3}}$$

COMSOL Multiphysics calculates the first Piola-Kirchhoff stress $\boldsymbol{P}$ by symbolic differentiation of the strain energy expression.

The hyperelastic material models directly supported are:

Neo-Hookean

$$W_s = \frac{1}{2}\mu(\bar{I}_1 - 3) + \frac{1}{2}\kappa(J_{\mathrm{el}} - 1)^2 \tag{7-4}$$

Mooney-Rivlin

$$W_s = C_{10}(\bar{I}_1 - 3) + C_{01}(\bar{I}_2 - 3) + \frac{1}{2}\kappa(J_{\mathrm{el}} - 1)^2 \tag{7-5}$$

The quantity J_{el} is defined in Equation 7-9 below.

Instead of the normal approach—using the conjugate pair formed by the second Piola-Kirchhoff stress and the Green-Lagrange strain—use the first Piola-Kirchhoff stress, P, and its conjugate strain, the displacement gradient, $\nabla\mathbf{u}$. This makes it possible to utilize the capability of COMSOL Multiphysics to automatically differentiate an expression, thus making it easy to modify only the strain energy function. The first Piola-Kirchhoff stresses are calculated as

$$P = \frac{\partial W_s}{\partial \nabla \mathbf{u}}$$

The variation of the energy can then be expressed as

$$\sum_{i,j}\left(\frac{\partial u_i}{\partial x_j}\right)_{\mathrm{test}} P_{ij}$$

Materials that are nearly incompressible cannot be solved using only displacement variables. The remedy is to introduce the pressure, p, as a dependent variable. The hyperelastic material model supports both the normal displacement-based formulation and the so-called mixed formulation that includes the pressure. The pressure is related to the volume change through the relation

$$p = -\kappa(J_{\mathrm{el}} - 1) \tag{7-6}$$

where κ is the bulk modulus.

The energy equations where the pressure is a dependent variable are:

Neo-Hookean

$$W_s = \frac{1}{2}\mu(\bar{I}_1 - 3) - p(J_{\mathrm{el}} - 1) - \frac{p^2}{2\kappa} \tag{7-7}$$

Mooney-Rivlin

$$W_s = C_{10}(\bar{I}_1 - 3) + C_{01}(\bar{I}_2 - 3) - p(J_{\mathrm{el}} - 1) - \frac{p^2}{2\kappa} \tag{7-8}$$

It can be shown that these equations results in the same energy and gives the correct contributions to the displacement and pressure equations as Equation 7-4 and Equation 7-5.

The second Piola-Kirchhoff stress, S, and the Cauchy stress, σ, can then be calculated from the first Piola-Kirchhoff stress:

$$S = F^{-1}P$$

$$\sigma = J^{-1}PF^{T}$$

Thermal Expansion

If thermal expansion is present, a stress-free volume change occurs. In this case, J_{el} in the constitutive relations above must be regarded as the elastic part of the total volume change, that is

$$J_{\mathrm{el}} = \frac{J}{J_{\mathrm{th}}} = \frac{J}{(1+\varepsilon_{\mathrm{th}})^3} \tag{7-9}$$

Thermal Strain

Thermal strain depends on the present temperature, T, the stress-free reference temperature, T_{ref}, and the thermal expansion vector, α_{vec}

$$\varepsilon_{\mathrm{th}} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix}_{\mathrm{th}} = \alpha_{\mathrm{vec}}(T - T_{\mathrm{ref}})$$

Depending on the material model, you set up α_{vec} up differently: For an isotropic material

$$\alpha_{\mathrm{vec}} = \begin{bmatrix} \alpha \\ \alpha \\ \alpha \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

for an orthotropic material

$$\alpha_{\mathrm{vec}} = \begin{bmatrix} \alpha_x \\ \alpha_y \\ \alpha_z \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

and for an anisotropic material you enter the full thermal expansion vector as input:

$$\alpha_{\mathrm{vec}} = \begin{bmatrix} \alpha_x \\ \alpha_y \\ \alpha_z \\ 2\alpha_{xy} \\ 2\alpha_{yz} \\ 2\alpha_{xz} \end{bmatrix}$$

ENTROPY

For a linear thermoelastic solid, the entropy per unit volume is:

$$S = \rho C_P \log(T/T_0) + S_{\mathrm{elast}}$$

where T_0 is the reference temperature, the volumetric heat capacity ρC_P is assumed independent of the temperature, and

$$S_{\mathrm{elast}} = \alpha_{\mathrm{vec}} \cdot \sigma \qquad (7\text{-}10)$$

where σ is the stress vector, and α_{vec} is the thermal expansion vector. For an isotropic material, Equation 7-10 simplifies into

$$S_{\mathrm{elast}} = \alpha(\sigma_x + \sigma_y + \sigma_z) \tag{7-11}$$

The entropy is a function of state and thus independent of the strain rate. The stress vector σ in the definitions Equation 7-10 and Equation 7-11 corresponds to no damping when used in a frequency response analysis, because the damping represents the rate-dependent (viscoelastic) effects in the material.

If you model the damping in the structural analysis via the loss factor, use the following definition for the elastic part of the entropy:

$$S_{\mathrm{elast}} = \alpha_{\mathrm{vec}} \cdot (\sigma - j\eta D\varepsilon)$$

where η is the loss factor, and j is the imaginary unit. For more details, see "Loss Factor Damping" on page 122, and the example "Heat Generation in a Vibrating Structure" on page 703 of the *Structural Mechanics Module Model Library.*

Initial Stress and Strain

Initial stress refers to the stress before the system applies any loads, displacements, or initial strains, written as

$$\sigma_0 = \begin{bmatrix} \sigma_{x0} \\ \sigma_{y0} \\ \sigma_{z0} \\ \tau_{xy0} \\ \tau_{yz0} \\ \tau_{xz0} \end{bmatrix}$$

The initial strain is the one before the system has applied any loads, displacements, or initial stresses

$$\varepsilon_0 = \begin{bmatrix} \varepsilon_{x0} \\ \varepsilon_{y0} \\ \varepsilon_{z0} \\ 2\varepsilon_{xy0} \\ 2\varepsilon_{yz0} \\ 2\varepsilon_{xz0} \end{bmatrix}$$

Follower Loads

Follower loads are loads that change direction as the body deforms. The most common type of follower load is a pressure acting on a surface. In this case the force changes size due to the stretching of the surface and direction due to the change in normal direction. The following section only considers this type of follower loads.

THEORY

The continuum application modes are formulated in the reference frame (the default). All forces must be specified as force/undeformed area in a fixed coordinate system (not in a system following the body). This makes it difficult to model a pressure acting on a surface because the force changes direction due to the deformation. There is also an area effect due to the stretching of the surface. The relation between the deformed area da and the undeformed area dA is needed. To handle this, the software uses a deformed frame that computes both the deformed area and the deformed normal direction ($\mathbf{n}$). The force is calculated as

$$\mathbf{F}dA = -P\mathbf{n}da = -P\mathbf{n}\frac{da}{dA}dA$$

where P are the follower pressure.

Plane Stress

In a plane stress condition the out-of-plane deformation causes the thickness to change, and this area effect is included explicitly. The equation transforms to

$$\mathbf{F}dA = -P\mathbf{n}\frac{da}{dA}dA\left(1+\frac{\partial w}{\partial z}\right)$$

Axial Symmetry

The extra r in the circumferential integration of the force expressions is transformed to $r + uaxi$ to account for the deformation.

Implementation

The COMSOL Multiphysics implementation of these equations in the application modes for structural analysis is based on the principle of virtual work expressed in global or local stress and strain components. The principle of virtual work states that the sum of virtual work from internal strains is equal to work from external loads.

The total stored energy, W, for a linear material from external and internal strains and loads equals

$$W = \int_V \left(\frac{1}{2}(-\varepsilon_x\sigma_x - \varepsilon_y\sigma_y - \varepsilon_z\sigma_z - 2\varepsilon_{xy}\tau_{xy} - 2\varepsilon_{yz}\tau_{yz} - 2\varepsilon_{xz}\tau_{xz}) + \mathbf{u}^t\mathbf{F}_V\right)dv$$

$$+ \int_S \mathbf{u}^t\mathbf{F}_S ds + \int_L \mathbf{u}^t\mathbf{F}_L dl + \sum_p \mathbf{U}^t\mathbf{F}_P .$$

The principle of virtual work states that

$$\delta W = 0 ,$$

and in order to derive the expression for the variation of W you differentiate symbolically, reaching the expression

$$\delta W = \int_V (-\varepsilon_{x\mathrm{test}}\sigma_x - \varepsilon_{y\mathrm{test}}\sigma_y - \varepsilon_{z\mathrm{test}}\sigma_z$$

$$-2\varepsilon_{xy\mathrm{test}}\tau_{xy} - 2\varepsilon_{yz\mathrm{test}}\tau_{yz} - 2\varepsilon_{xz\mathrm{test}}\tau_{xz} + \mathbf{u}^t_{\mathrm{test}}\mathbf{F}_V)dv$$

$$+ \int_S \mathbf{u}^t_{\mathrm{test}}\mathbf{F}_S ds + \int_L \mathbf{u}^t_{\mathrm{test}}\mathbf{F}_L dl + \mathbf{U}^t_{\mathrm{test}}\mathbf{F}_P .$$

The principle of virtual work for the axial symmetry case reads

$$\delta W = \int_A r(-\varepsilon_{r\mathrm{test}}\sigma_r - \varepsilon_{\varphi\mathrm{test}}\sigma_\varphi - \varepsilon_{z\mathrm{test}}\sigma_z - 2\varepsilon_{rz\mathrm{test}}\tau_{rz}$$

$$+ r \cdot \mathrm{uor}_{\mathrm{test}}F_r + w_{\mathrm{test}}F_z)dA +$$

$$\int_S r(r \cdot \mathrm{uor}_{\mathrm{test}}F_r + w_{\mathrm{test}}F_z)ds + (r \cdot \mathrm{uor}_{\mathrm{test}}F_r + w_{\mathrm{test}}F_z\ )/2\pi = 0$$

To avoid division by r, the true radial displacement, u is replaced in the above equation by a new dependent variable

$$\mathrm{uor} = \frac{u}{r} .$$

If you define the material in a local user-defined coordinate system, the variational equation in COMSOL Multiphysics is expressed in local instead of global stresses and strains.

To create the strain tensor in local coordinates, transform the global strain tensor

$$\varepsilon_1 = T^T \varepsilon_g T$$

where T is the local-to-global coordinate-system transformation matrix.

Then calculate the local stress tensor from the local strain, and the global stress tensor by transforming the local stress tensor

$$\sigma_g = T \sigma_1 T^T$$

SETTING UP EQUATIONS FOR DIFFERENT ANALYSES

All application modes in the Structural Mechanics Module support static, eigenfrequency, transient, frequency-response, parametric, and quasi-static transient analyses. Each type might solve a different equation or employ a different solver. You control this choice with the **Analysis type** property that appear in the **Application Mode Properties** dialog box for the corresponding application mode.

Static, Parametric, and Quasi-Static Transient Analysis

These analyses all use the same equation, the difference being what solver that is used. In the following, static analysis is used as short for all the above analyses because they use the same equations.

COMSOL Multiphysics' implementation is based on the stress and strain variables. The normal and shear strain variables depend on the displacement derivatives (described in general 3D terms in the section "Theory Background" on page 164); the normal and shear stress variables depend on the strains (described in general 3D terms in the section"Stress-Strain Relationship" on page 166).

Using the shear and stress variables, you can express the principle of virtual work as

$$\begin{aligned}\delta W = \int_V &(-\varepsilon_{x\mathrm{test}}\sigma_x - \varepsilon_{y\mathrm{test}}\sigma_y - \varepsilon_{z\mathrm{test}}\sigma_z \\ &-2\varepsilon_{xy\mathrm{test}}\tau_{xy} - 2\varepsilon_{yz\mathrm{test}}\tau_{yz} - 2\varepsilon_{xz\mathrm{test}}\tau_{xz} + \mathbf{u}_{\mathrm{test}}^T\mathbf{F}_V)dv \\ &+ \int_S \mathbf{u}_{\mathrm{test}}^T\mathbf{F}_S ds + \int_L \mathbf{u}_{\mathrm{test}}^T\mathbf{F}_L dl + \sum_p U_{\mathrm{test}}^T\mathbf{F}_P = 0\end{aligned}$$

If you describe the material in a local coordinate system, δW is expressed in local stresses and strains.

Transient Analysis

For transient problems consider Newton's second law

$$\rho\frac{\partial^2\mathbf{u}}{\partial t^2} - \nabla\cdot c\nabla\mathbf{u} = \mathbf{F}.$$

It defines the equation of motion with *no damping*.

To model viscous damping, COMSOL Multiphysics uses *Rayleigh damping*, where you specify two damping coefficients. As an example, consider a system with a single degree of freedom. The equation of motion for such a system with viscous damping is

$$m\frac{d^2u}{dt^2} + \xi\frac{du}{dt} + ku = f(t).$$

In the Rayleigh damping model, you express the damping parameter ξ in terms of the mass m and the stiffness k as

$$\xi = \alpha_{dM}m + \beta_{dK}k$$

The Rayleigh damping proportional to mass and stiffness is added to the static weak term.

Frequency Response Analysis

You specify harmonic loads using two components:

- The amplitude value, F_x
- The phase, $F_{x\mathrm{Ph}}$

To derive the equations for the steady-state response from harmonic excitation loads

$$F_{x\mathrm{freq}} = F_x(f)\cdot\cos\left(\omega t + F_{x\mathrm{Ph}}(f)\frac{\pi}{180}\right)$$

$$\mathbf{F}_{\mathrm{freq}} = \begin{bmatrix} F_{x\mathrm{freq}} \\ F_{y\mathrm{freq}} \\ F_{z\mathrm{freq}} \end{bmatrix},$$

assume a harmonic response with the same angular frequency as the excitation load

$$u = u_{\mathrm{amp}}\cos(\omega t + \phi_u)$$

$$\mathbf{u} = \begin{bmatrix} u \\ v \\ w \end{bmatrix}$$

You can also describe this relationship using complex notation

$$u = \mathrm{Re}(u_{\mathrm{amp}} e^{j\phi_u} e^{j\omega t}) = \mathrm{Re}(\tilde{u} e^{j\omega t}) \text{ where } \tilde{u} = u_{\mathrm{amp}} e^{j\phi_u}$$

$$\mathbf{u} = \mathrm{Re}(\tilde{\mathbf{u}} e^{j\omega t})$$

$$F_{x\mathrm{freq}} = \mathrm{Re}\left(F_x(\omega) e^{jF_{xPh}(f)\frac{\pi}{180}} e^{j\omega t}\right) = \mathrm{Re}(\tilde{F}_x e^{j\omega t})$$

where

$$\tilde{F}_x = F_x(f) e^{jF_{xPh}(f)\frac{\pi}{180}}$$

$$\tilde{\mathbf{F}} = \begin{bmatrix} \tilde{F}_x \\ \tilde{F}_y \\ \tilde{F}_z \end{bmatrix}$$

Eigenfrequency Analysis

The eigenfrequency equations are derived by assuming a harmonic displacement field, similar as for the frequency response formulation. The difference is that this analysis type uses a new variable $j\omega$ explicitly expressed in the eigenvalue.

$$j\omega = -\lambda$$

The eigenfrequency f is then derived from $j\omega$ as

$$f = \left|\frac{\mathrm{Im}(j\omega)}{2\pi}\right|$$

In the eigenfrequency analysis no damping is added to the equations.

Damped Eigenfrequency Analysis

This analysis type is similar to the eigenfrequency analysis except that it adds viscous damping terms to the equation. The analysis type supports Rayleigh damping. In addition to the eigenfrequency you can also look at the quality factor, Q, and decay factor, δ, for the model:

$$Q = \frac{\mathrm{Im}(\lambda)}{2\mathrm{Re}(\lambda)}$$

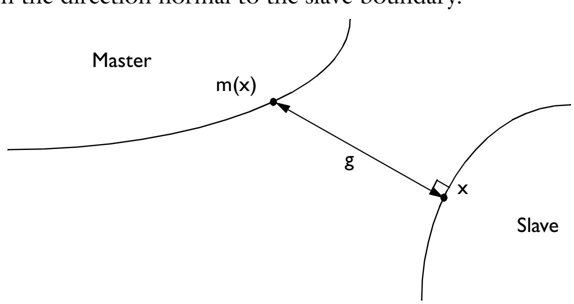
$$\delta = \mathrm{Re}(\lambda)$$

Contact Modeling

COMSOL Multiphysics solves contact problems using an *augmented Lagrangian* method. This means that the software solves the system in a segregated way. Augmentation components are introduced for the contact pressure T_n and the components T_{ti} of the friction traction vector T_t. An additional iteration level is added where the usual displacement variables are solved separately from the contact pressure and traction variables. The algorithm repeats this procedure until it fulfills a convergence criterion.

In the following equations F is the deformation gradient matrix. When looking at expressions evaluated on the slave boundaries, the expression $\mathrm{map}(E)$ denotes the value of the expression E evaluated at a corresponding master point, and g is the gap distance between the slave and master boundary.

Both the contact map operator $\mathrm{map}(E)$ and the gap distance variable are defined by the contact element `elcontact` (see the documentation of `elcontact` on page 55 of the *COMSOL Multiphysics Reference Guide* for details). For each slave point where the operator or gap is evaluated, a corresponding master point is sought by searching in the direction normal to the slave boundary.



Note that before the boundaries come in contact, the master point found is not necessarily the point on the master boundary closest to the slave point. However, as the boundaries approach one another, the master point converges to the closest point as the gap distance goes to zero.

Using the special gap distance variable, the penalized contact pressure T_{np} is defined on the slave boundary as

$$T_{\mathrm{np}} = \begin{cases} T_{\mathrm{n}} - p_{\mathrm{n}} g & \text{if } \ g \le 0 \\ T_{\mathrm{n}} e^{-\frac{p_n g}{T_n}} & \text{otherwise} \end{cases} \tag{7-12}$$

where g is the gap distance between the *slave* and *master* boundary, and p_{n} is the user defined normal penalty factor.

The *penalized friction traction* T_{tp} is defined on the slave boundary as:

$$T_{\mathrm{tp}} = \min\left(\frac{T_{\mathrm{tcrit}}}{|T_{\mathrm{ttrial}}|}, 1\right) T_{\mathrm{ttrial}} \tag{7-13}$$

where T_{ttrial} is defined as

$$T_{\mathrm{ttrial}} = T_{\mathrm{t}} - p_{\mathrm{t}} \mathrm{map}(F)(x^m - x^m{}_{\mathrm{old}}) \tag{7-14}$$

and

$$x^m = \mathrm{map}(x) \tag{7-15}$$

where x are the space coordinates.

In Equation 7-14 p_{t} is the user-defined friction traction penalty factor, and $x^m{}_{\mathrm{old}}$ is the value of x^m in the last time step, and

$$\mathrm{map}(F)(x^m - x^m{}_{\mathrm{old}}) \tag{7-16}$$

is the vector of slip since the last time step (approximated using a backward Euler step).

T_{tcrit} is defined as

$$T_{\mathrm{tcrit}} = \min(\mu T_{\mathrm{np}} + \mathrm{cohe}, T_{\mathrm{tmax}}) \tag{7-17}$$

In Equation 7-17 μ is the friction coefficient, cohe is the user-defined cohesion sliding resistance, and T_{tmax} is the user-defined maximum friction traction.

In the following equation δ is the variation (represented by the test operator in COMSOL Multiphysics). The contact interaction gives the following contribution to the weak equation on the slave boundary:

$$\int_{\text{slave}} (T_{\text{np}}\delta g + T_{\text{tp}} \cdot m(F)\delta x^{m})dA + \int_{\text{slave}} (w_{\text{cn}}\delta T_{\text{n}} + w_{\text{ct}} \cdot \delta T_{\text{t}})dA \qquad (7\text{-}18)$$

where w_{cn} and w_{ct} are contact help variables defined as:

$$w_{\text{cn}} = T_{\text{np},\,i} - T_{\text{n},\,i+1} \qquad (7\text{-}19)$$

$$w_{\text{ct}} = (\text{friction}\ \ (T_{\text{tp}} - (\mathbf{n} \cdot T_{\text{tp}})n))_{i} - T_{\text{t},\,i+1} \qquad (7\text{-}20)$$

where i is the augmented solver iteration number and friction is a Boolean variable stating if the parts are in contact.

FRICTION

The friction model is either no friction or Coulomb friction.

The friction coefficient μ is defined as

$$\begin{cases} \mu_{\text{d}} + (\mu_{\text{s}} - \mu_{\text{d}})e^{-\text{dcfric}|v_s|} & \text{if dynamic friction} \\ \mu_{\text{s}} & \text{otherwise} \end{cases} \qquad (7\text{-}21)$$

where μ_{s} is the static coefficient of friction and μ_{d} is the *dynamic friction coefficient.* v_{s} is the slip velocity, and dcfric is a decay coefficient.

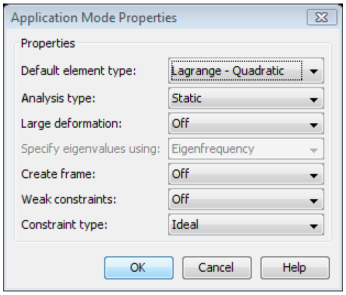
Application Mode Description

This section describes how to define a model using the continuum application modes in the Structural Mechanics Module. It reviews the following subsections:

- Application mode properties
- Scalar variables
- Material
- Constraints
- Loads
- Thermal coupling
- Damping
- Initial stresses and strains
- Perfectly matched layers (PMLs)
- Contact

Properties

To set or examine application mode properties, go to the **Physics>Properties** menu to open the **Application Mode Properties** dialog box. Here you control various global settings for the model:



Application Mode Properties dialog box for the structural mechanics application modes.

- **Default element type**: The selected finite element type that makes up the discretized finite element model is the default on all new subdomains, and the choice does not affect subdomains already created. Available elements are:
  - **Lagrange - Linear**
  - **Lagrange - Quadratic**
  - **Lagrange - Cubic**
  - **Lagrange - Quartic**
  - **Lagrange - Quintic**
  - **Lagrange -** $\mathbf{U_2P_1}$
  - **Lagrange -** $\mathbf{U_3P_2}$
  - **Lagrange -** $\mathbf{U_4P_3}$
  - **Lagrange -** $\mathbf{U_5P_4}$
- **Analysis type**: This drop-down list shows the various analyses you can perform; the default is **Static**. Your choice affects both the equations and which solver COMSOL Multiphysics uses when the **Auto select solver** option in the **Solver Parameters** dialog box is active.

ANALYSIS TYPE	COMSOL MULTIPHYSICS SOLVER
Static	Stationary
Static elasto-plastic material	Parametric

ANALYSIS TYPE	COMSOL MULTIPHYSICS SOLVER
Eigenfrequency	Eigenvalue
Damped Eigenfrequency	Eigenvalue
Time dependent	Time dependent
Frequency response	Parametric
Parametric	Parametric
Quasi-static transient	Time dependent
Linear buckling	Eigenvalue

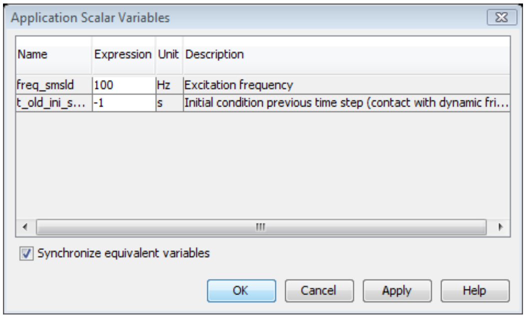
- **Large deformation**: This list controls whether or not the model should include large deformations, which only the Principle of virtual work supports (see next item). The default is **Off**.
- **Specify eigenvalues using**: This list controls how to work with eigenmode analyses. Here you should specify **Eigenvalue** or **Eigenfrequency**; this property is enabled only for eigenfrequency and linear-buckling analyses.
- **Create frame**: This list controls whether or not to create a deformed frame. The default is **Off**. A deformed frame is needed in contact modeling and to define follower forces but can also be used in a multiphysics context to define some other physics on.
- **Eigenfrequency**; this property is enabled only for eigenfrequency, damped eigenfrequency and linear-buckling analyses.
- **Weak constraints**: Controls whether or not weak constraints are active. Use weak constraints for accurate reaction-force computation. When weak constraints are enabled, all constraints are weak by default, but it is possible to change this setting for individual domains.
- **Constraint type**: Constraints can be ideal or nonideal (see "Ideal vs. Non-Ideal Constraints" on page 301 in the *COMSOL Multiphysics Modeling Guide*).

Scalar Variables

There are three different scalar variables:

- Excitation frequency, `freq`, which is applicable only for frequency response analysis.

- Initial condition for the time in the previous time step, `t_old_ini`, which is applicable only for contact modeling using dynamic friction.
- Complex angular frequency, `jomega`, which is applicable only for eigenfrequency analysis. You normally do not need to edit the complex angular frequency.



The Application Scalar Variables dialog box in a frequency response analysis.

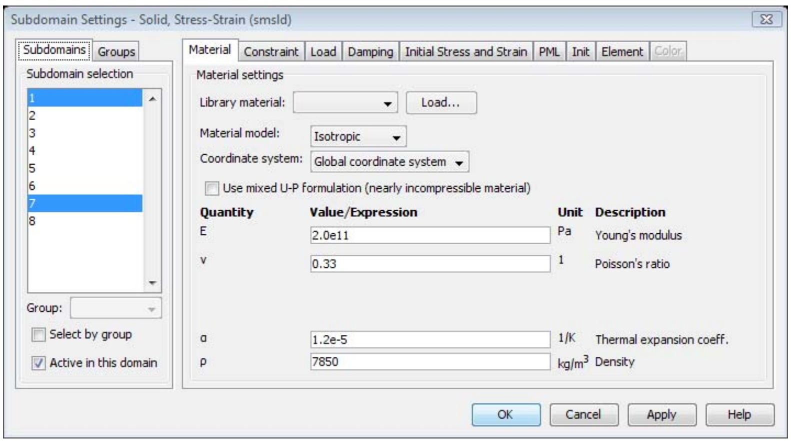
When you select a frequency response analysis, the parametric solver becomes the default solver, which makes it easy to perform a frequency sweep over several excitation frequencies in one analysis. In this case, choose **Solve>Solver Parameters**, and in the dialog box that appears go to the **General** page. In the **Parameter** area, enter `freq_smsld` in the **Parameter name** edit field. Values that you enter in the **Parameter values** edit field override the excitation frequency you might have entered in the **Application Scalar Variables** dialog box.

To access the excitation frequency f use the variable `freq` and to access the angular excitation frequency ω use `omega`.

Material Properties

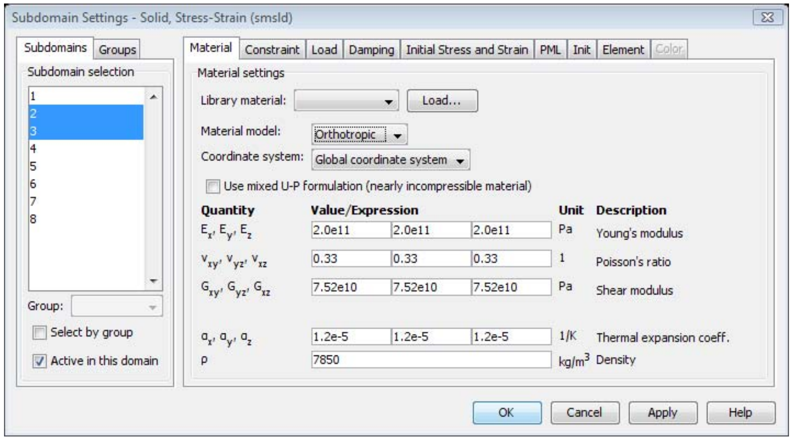
You define material properties on the **Material** page in the **Subdomain Settings** dialog box. This page has two lists: **Material model** and **Coordinate system**. Now consider the options available for each of these lists:

- **Material model**: When you select the type of material, a set of appropriate material properties appear in the dialog box.
  - **Isotropic**: This material has the same properties in all directions.



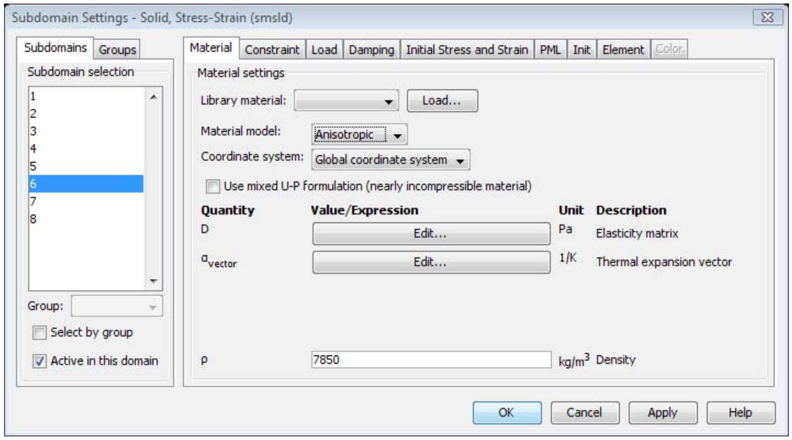
Material properties for an isotropic material.

  - **Orthotropic**: This material has different material properties in different directions, and its stiffness depends on the properties E_i, ν_{ij}, and G_{ij} (see page 166 for details). In addition, thermal expansion depends on the parameter α_i (see page 178 for details).

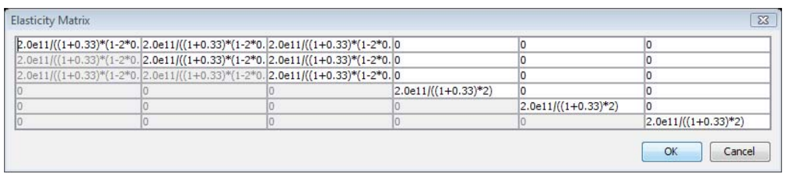


Material properties for an orthotropic material.

- **Anisotropic**: This material has different material properties in different directions, and the stiffness comes from the symmetric *elasticity matrix*, D (see page 166 for details). Thermal expansion depends on the *thermal expansion vector*, α_{vec} (see page 178 for details).

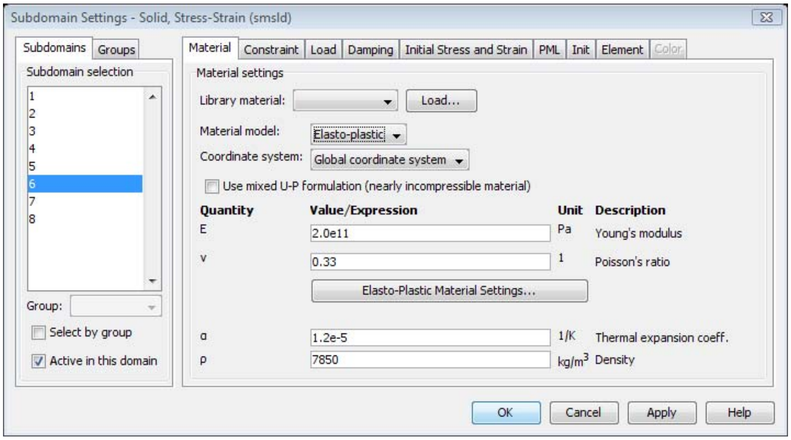


Material properties for an anisotropic material.

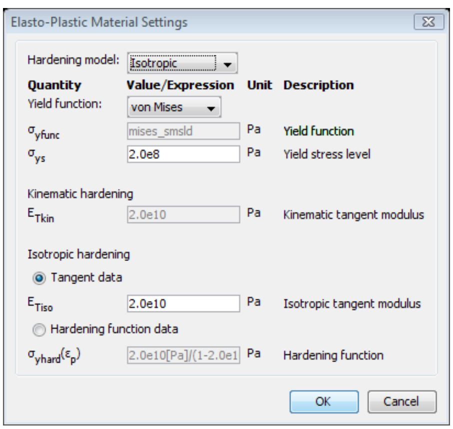


The Elasticity Matrix dialog box for entering the components of the D matrix for an anisotropic material.

- **Elasto-plastic**: A nonlinear material with possible hardening (see page 173 for details).

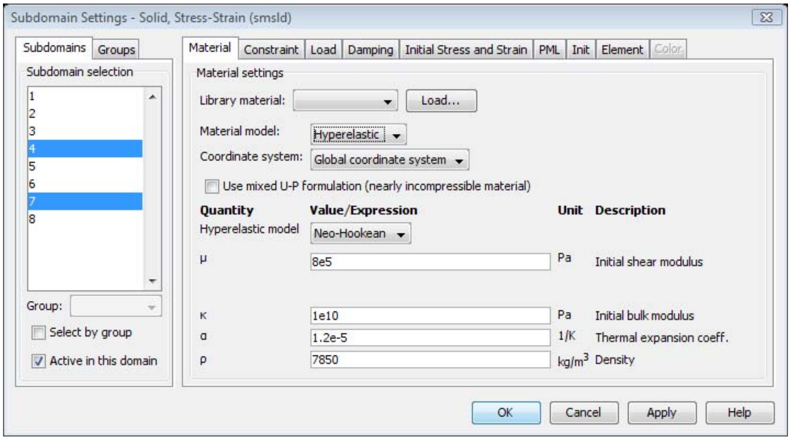


Material properties for an elasto-plastic material.



The Elasto-Plastic Material Settings dialog box for specification of elasto-plastic material data.

- **Hyperelastic**: A hyperelastic material based on a strain energy density function, often used to model rubberlike materials (see page 176 for details).



Material properties for a hyperelastic material.

- **Coordinate system**: In this second list on the **Material** page you select the coordinate system on which the material properties are defined. Use it for orthotropic and anisotropic materials that are defined in another coordinate system other than the global system or if you need stresses and strains in a local coordinate system for postprocessing. The **Coordinate system** list is disabled if no user-defined coordinate systems are available. To open the **Coordinate System Settings** dialog box, go to the **Options** menu and choose **Coordinate Systems**.
- **Use mixed U-P formulation (nearly incompressible material)**: Controls whether to use a mixed formulation adding the negative mean pressure as a dependent variable to solve for. This can also be controlled from the **Predefined element** list on the **Element** page. Nearly incompressible materials means a Poisson's ratio close to 0.5. See page 170 for details.

TABLE 7-1: MATERIAL PROPERTIES FOR VARIOUS MATERIAL MODELS

PARAMETER	VARIABLE	DESCRIPTION	MATERIAL MODEL
E	E	Young's modulus	Isotropic/ elasto-plastic
ν	nu	Poisson's ratio	Isotropic/ elasto-plastic
ρ	rho	Density	All
α	alpha	Thermal-expansion coefficient	Isotropic
th	`thickness`	The thickness of the geometry	All

TABLE 7-1: MATERIAL PROPERTIES FOR VARIOUS MATERIAL MODELS

PARAMETER	VARIABLE	DESCRIPTION	MATERIAL MODEL
E_i	E`i`	Young's modulus in the x_i direction	Orthotropic
ν_{ij}	nu`ij`	Poisson's ratio for the $x_i x_j$-plane	Orthotropic
G_{ij}	G`ij`	Shear modulus for the $x_i x_j$-plane	Orthotropic
α_i	alpha`i`	Thermal-expansion coefficient in the x_i direction	Orthotropic
D		Elasticity matrix for the anisotropic case	Anisotropic
α_{vec}		Thermal-expansion vector for the anisotropic case	Anisotropic
σ_{ys}	Sys	Yield stress level	Elasto-plastic
σ_{yfunc}	Syfunc	Yield function	Elasto-plastic
σ_{yhard}	Syhard	Hardening function for isotropic hardening	Elasto-plastic
E_{Tiso}	ETiso	Isotropic-tangent modulus	Elasto-plastic
E_{Tkin}	ETkin	Kinematic-tangent modulus	Elasto-plastic
C_{10}	C10	Mooney-Rivlin material parameter	Hyperelastic
C_{01}	C01	Mooney-Rivlin material parameter	Hyperelastic
μ	mu	Initial shear modulus	Hyperelastic
κ	kappa	Initial bulk modulus	Hyperelastic

The index i in the parameters E_i and α_i in Table 7-1 refers to the space coordinates x_i and represents the following names for the different application modes:

APPLICATION MODE	x_1	x_2	x_3
Plane Stress and Plane Strain	x	y	z
Solid	x	y	z
Axisymmetry Stress-strain	r	φ	z

Example: E_i for axisymmetry stress-strain means E_r, E_φ , and E_z.

The parameter ν_{ij} in Table 7-1 refers to the space coordinates $x_i x_j$ and is defined for the following combinations of i and j for the different application modes:

APPLICATION MODE	x_1x_2	x_2x_3	x_1x_3
Plane Stress and Plane Strain	xy	yz	xz
Solid	xy	yz	xz
Axisymmetry Stress-strain	$r\varphi$	φz	rz

and the parameter G_{ij} is defined for these combinations:

APPLICATION MODE	x_1x_2	x_2x_3	x_1x_3
Plane Stress and Plane Strain	xy		
Solid	xy	yz	xz
Axisymmetry Stress-strain			rz

Note: You can change the default names for the space coordinates in the same way as you can the names of the dependent variables.

Now examine the various material properties in Table 7-1.

Young's modulus It defines a material's modulus of elasticity, E. For an isotropic material it is the spring stiffness in Hooke's law, which in 1D form is

$$\sigma = E\varepsilon$$

where σ is the stress and ε is the strain. An orthotropic material uses one value of Young's modulus for each direction, E_i as defined on page 166.

Poisson's ratio Denoted by ν, it defines the normal strain in the perpendicular direction, generated from a normal strain in the other direction and follows the equation

$$\varepsilon_{\perp} = -\upsilon\varepsilon_{||} .$$

An orthotropic material defines three values of ν_{ij}.

Note: ν_{ij} is defined differently depending on the application field, so review page 166 for the definition within COMSOL Multiphysics. It is easy to transform among definitions, but you must check which one your material uses.

Shear Modulus Denoted by G_{ij}, it defines the relationship between engineering shear strain and shear stress. It is applicable only to an orthotropic material and follows the equation

$$\varepsilon_{ij} = \frac{\tau_{ij}}{G_{ij}}.$$

Density This entry specifies ρ, the material's density.

Thermal expansion coefficient It defines how much a material expands due to an increase in temperature following the equation

$$\varepsilon_{\mathrm{th}} = \alpha(T - T_{\mathrm{ref}})$$

where $\varepsilon_{\mathrm{th}}$ is the thermal strain, and α is the thermal expansion coefficient. With it you model thermal strain for an isotropic material. For an orthotropic material, three values of α_i are defined for the three perpendicular directions.

Thickness (th) This property defines the out of plane thickness of the geometry for the Plane Stress and Plane Strain application modes.

Elasticity matrix It defines the elasticity matrix, D, for anisotropic materials (see page 167 for details). For the Plane Stress and Plane Strain application modes D is defined as a 4-by-4 matrix, since the out of plane shear stress and shear strain components are zero.

Thermal expansion vector It defines the thermal expansion vector, α_{vec}, for anisotropic materials (see page 178 for details).

Yield stress level (σ_{ys}) This parameter gives the stress level where plastic deformation starts. In the theory section this parameter is named Y_0.

Yield function (σ_{func}) This function detects if plasticity has occurred. In the theory section this parameter is named ϕ .

Isotropic tangent modulus This parameter is the tangent modulus used for isotropic hardening. This parameter together with σ_{ys} defines the f_2 function from the theory section as

$$f_2(\varepsilon_{\mathrm{pe}}) = \sigma_{\mathrm{ys}} + \frac{E_{T\mathrm{iso}}}{1 - \frac{E_{T\mathrm{iso}}}{E}}\varepsilon_{\mathrm{pe}}$$

Kinematic tangent modulus This parameter is the tangent modulus used for kinematic hardening. This parameter is used to calculate the σ_{shift} parameter from the theory section as

$$\sigma_{\mathrm{shift}} = \frac{E_{T\mathrm{kin}}}{1 - \frac{E_{T\mathrm{kin}}}{E}} \cdot \frac{2}{3} \cdot \varepsilon_p$$

Hardening function (σ_{yhard}) This hardening function applies to isotropic hardening. This parameter together with σ_{ys} defines the f_2 function from the theory section as

$$f_2(\varepsilon_{\mathrm{pe}}) = \sigma_{\mathrm{ys}} + \sigma_{\mathrm{yhard}}(\varepsilon_{\mathrm{pe}})$$

This definition implies that you have to subtract the yields stress level (σ_{ys}) when defining your hardening function.

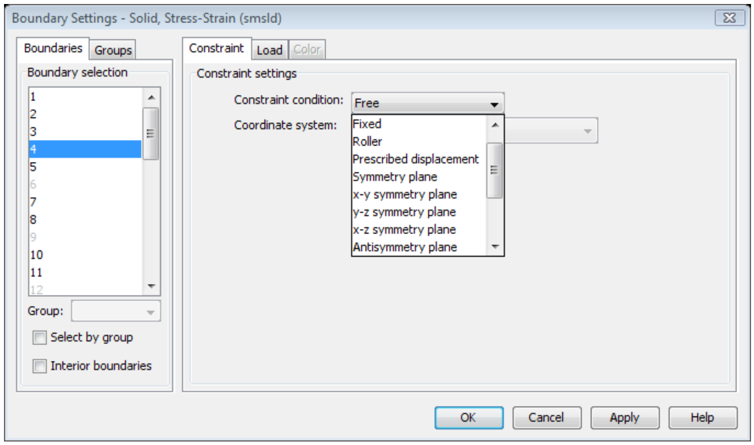
Mooney-Rivlin material parameters Hyperelastic material model parameters.

Initial shear modulus Used for Neo-Hookean hyperelastic material model.

Initial bulk modulus Used for Neo-Hookean and Mooney-Rivlin hyperelastic material models.

Constraints

A constraint specifies the displacement of certain parts of a structure. You can define constraints on all domain levels: points, edges, faces/boundaries, and subdomains in 3D; points, boundaries, and subdomains in 2D. To control a constraint, go to the **Constraint** page that appears in the following dialog boxes that you find in the **Physics** menu: **Physics>Subdomain Settings**, **Physics>Boundary Settings**, **Physics>Edge Settings**, and **Physics>Point Settings**. The following figure shows the **Boundary Settings** dialog box for the Solid, Stress-Strain application mode, but the **Constraints settings** area has the same appearance in all structural mechanics continuum application modes.



An example of a typical Constraint page, taken here from the Solid, Stress-Strain application mode Boundary Settings dialog box.

Within the dialog box the **Constraint condition** list lets you control what type of constraint you want to define. You can choose from the following options:

CONSTRAINT CONDITION	BOUNDARY	SUBDOMAIN	USE WHEN
Free	√	√	The domain has no constraint
Fixed	√	√	The displacement in the domain is fixed in all directions
Roller	√		The normal displacement is constrained
Prescribed displacement	√	√	The displacement in any direction need to be prescribed
Symmetry plane	√		The boundary is a symmetry plane
x-y symmetry plane	√		The selected coordinate system’s xy-plane is a symmetry plane
y-z symmetry plane	√		The selected coordinate system’s yz-plane is a symmetry plane
x-z symmetry plane	√		The selected coordinate system’s xz-plane is a symmetry plane
Antisymmetry plane	√		The boundary is an antisymmetry plane
x-y antisymmetry plane	√		The selected coordinate system’s xy-plane is an antisymmetry plane

CONSTRAINT CONDITION	BOUNDARY	SUBDOMAIN	USE WHEN
y-z antisymmetry plane	√		The selected coordinate system's yz-plane is an antisymmetry plane
x-z antisymmetry plane	√		The selected coordinate system's xz-plane is an antisymmetry plane
Prescribed velocity	√	√	The velocity in any direction need to be prescribed, only available for frequency response analysis
Prescribed acceleration	√	√	The acceleration in any direction need to be prescribed, only available for frequency response analysis

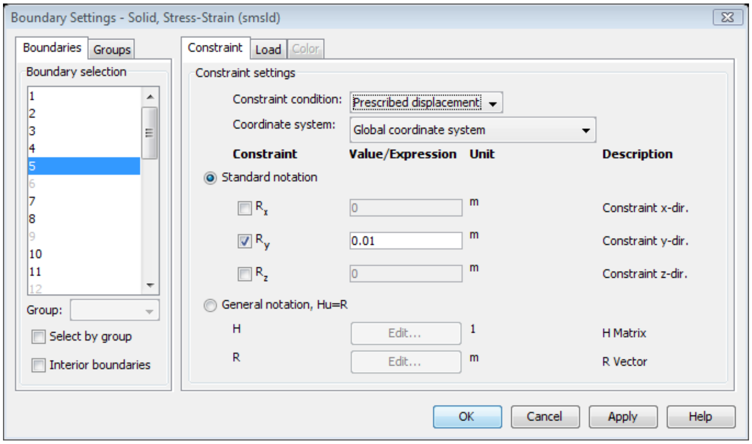
The symmetry or antisymmetry condition has the following interpretation.

CONDITION	X-DISPLACEMENT	Y-DISPLACEMENT	Z-DISPLACEMENT
x-y symmetry plane			√
y-z symmetry plane	√		
x-z symmetry plane		√	
x-y antisymmetry plane	√	√	
y-z antisymmetry plane		√	√
x-z antisymmetry plane	√		√

The **Coordinate system** list lets you control in which coordinate system you want the constraint defined. Available options are:

- Global coordinate system
- Tangent and normal coordinate system (available only on boundaries)
- User-defined coordinate systems if any local coordinate systems are defined.

When you select **Prescribed displacement** a number of new options appears in the dialog box, and the **Constraint** page takes on this appearance:



The Constraint page showing the Prescribed displacement options.

You can prescribe a constraint in two ways:

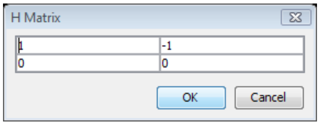
- In standard notation (select this option by clicking the **Standard notation** button), you constrain each displacement direction independently. The check boxes adjacent to the $\mathbf{R_x}$, $\mathbf{R_y}$, and $\mathbf{R_z}$ edit fields activate the constraint, whereupon you enter the value/expression of the displacement (the default value is `0`).
- In general notation (select this option by clicking the **General notation, Hu=R** button) lets you specify constraints as any linear combination of displacements components. For instance, in the 2D case, use the relationship

$$H\begin{bmatrix} u \\ v \end{bmatrix} = R .$$

  Enter values for the H matrix and R vector in corresponding dialog boxes by clicking the corresponding **Edit** buttons. For example, to achieve the condition $u = v$, use the settings

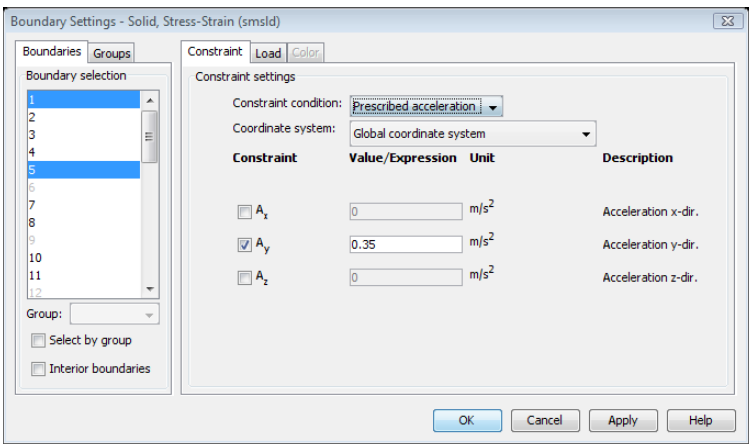
$$H = \begin{bmatrix} 1 & -1 \\ 0 & 0 \end{bmatrix}, \qquad R = \begin{bmatrix} 0 \\ 0 \end{bmatrix},$$

which force the domain to move only diagonally in the xy-plane.



The H Matrix dialog box for the example in the text.

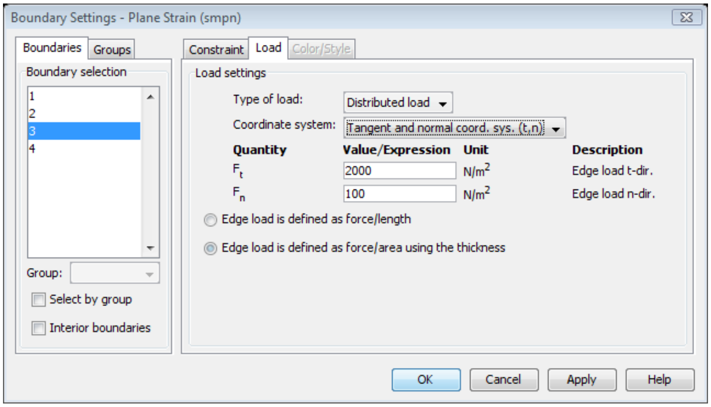
In a frequency response analysis you have the possibility to specify not only a harmonic displacement but also a harmonic velocity or acceleration. You specify a prescribed velocity or acceleration in the same way as **Prescribed displacement** using **Standard notation** by first selecting **Prescribed velocity** or **Prescribed acceleration** in the **Constraint condition** list.



Constraint page showing the Prescribed acceleration settings.

Loads

"Load" is a general term for a force applied to a structure. In the Structural Mechanics Module you can specify loads on all domain types using the **Load** page that appears in the following dialog boxes that you find on the **Physics** menu: **Subdomain Settings**, **Boundary Settings**, **Edge Settings**, and **Point Settings**.



The Boundary Settings dialog box for the Plane Strain application mode shown here is representative of load pages for all domain levels in all structural mechanics application modes.

The loads on all levels except the point level are given as *distributed loads* using a force density such as; force/length, force/area, or force/volume.

For boundaries you have the option to specify between different types of loads using the **Type of load** list. You select between distributed load and *follower load* (distributed load is the default setting).

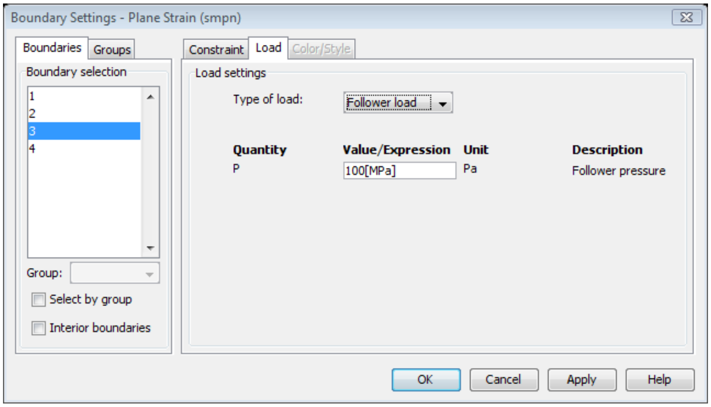
FOLLOWER LOADS

Follower loads are loads that depend on the deformation. The most common case is a pressure directed along the negative normal direction to a surface that deforms. In the following, follower loads imply such a load. Follower loads are only available on boundaries.

All loads must be applied in the undeformed reference frame; the software then computes the follower load using a frame that deforms with the structure. Both the direction and the size of the load change as the structure deforms. The **Create frame** application mode property is automatically set to **On** as soon as you specify a follower force.

Follower loads are only meaningful in a large deformation analysis. The **Large deformation** application mode property is automatically set to **On** as soon as you specify a follower force.

You select **Follower load** from the **Type of load** list on the **Load** page. You specify the pressure in the **P** edit field.



The Boundary Settings dialog box for the Plane Strain application mode showing the follower load setting.

DISTRIBUTED LOADS

Distributed load is the default setting on boundaries. On all other levels a distributed load is the only way to specify a load. For boundaries you select between distributed loads and follower loads using the **Type of load** list.

For plane stress and plane strain, two option buttons allow you to choose how to specify the load using the thickness. The following table shows how to define the loads on different domains in different application modes; the entries give the SI unit in parenthesis.

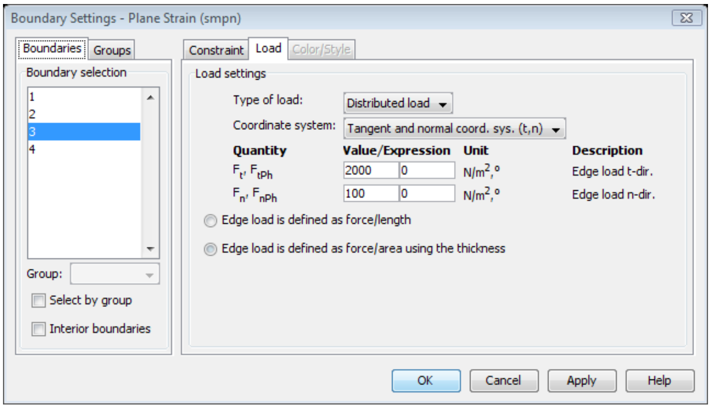
APPLICATION MODE	POINT	EDGE	BOUNDARY	SUBDOMAIN
Plane Stress, Plane Strain	force (N)		force/area (N/m^2) or force/length (N/m)	force/volume (N/m^3) or force/area (N/m^2)
Axisymmetry, Stress-Strain	total force along the circumferential (N)		force/area (N/m^2)	force/volume (N/m^3)
Solid, Stress-Strain	force (N)	force/length (N/m)	force/area (N/m^2)	force/volume (N/m^3)

Within the dialog box, the **Coordinate system** list lets you control in which coordinate system you want to define the load. Available options are:

- Global coordinate system

- Tangent and normal coordinate system (available only on boundaries)
- User-defined coordinate systems, if any local coordinate systems are defined

For a frequency response analysis you have additional input data. To control the analysis type, use the **Application Mode Properties** dialog box. When frequency response is the analysis type, the **Load** page takes on this appearance:



The Load page that appears for frequency response analysis.

For frequency response analysis, the application mode splits the harmonic load into two parameters:

- The amplitude, F
- The phase (F_{Ph})

Together they define a harmonic load whose amplitude and phase shift can vary with the excitation frequency, f

$$F_{\mathrm{freq}} = F(f) \cdot \cos(2\pi f + F_{\mathrm{Ph}}(f))\ .$$

For subdomains, you have additional options to control if and how the analysis should include thermal strains (explained in the following section).

Thermal Coupling

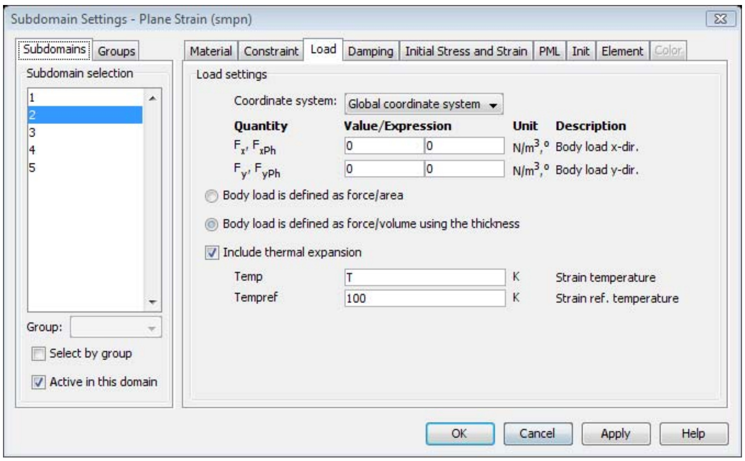
Material expands with temperature, causing thermal strains to develop in the material. The thermal strains, taken together with the initial strains and elastic strains from structural loads, form the total strain

$$\varepsilon = \varepsilon_{el} + \varepsilon_{th} + \varepsilon_0$$

where

$$\varepsilon_{th} = \alpha(T - T_{ref}) .$$

Thermal coupling means that the analysis includes thermal expansion. Details on thermal coupling appear on page 178. You specify thermal effects on the **Load** page in the dialog box that appears when you choose **Physics>Subdomain Settings**.



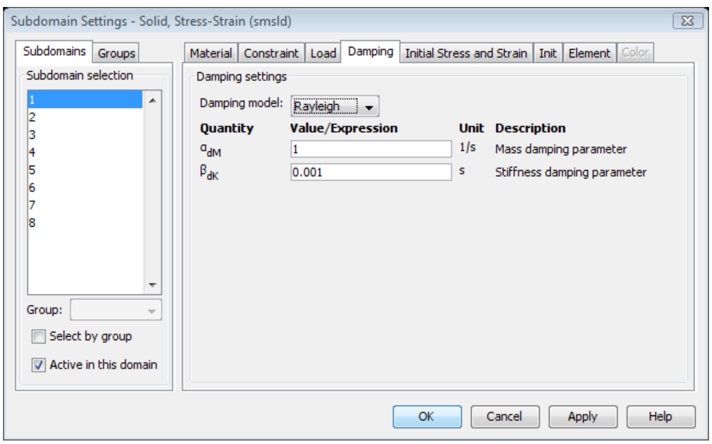
You specify thermal effects on the Load page.

The **Include thermal expansion** check box instructs the model to add thermal effects. Specify the strain temperature, T, and reference temperature, T_{ref}, in the **Temp** and **Tempref** edit fields, but you specify the thermal expansion coefficient on the **Material** page (see page 193). T and T_{ref} can be any expression and are typically another dependent variable for temperature solved for in a COMSOL Multiphysics heat transfer application mode. Any type of analysis can use this temperature coupling.

Note: Special approach is required if the structural analysis is performed in the frequency domain. This includes the following analysis types: **Frequency response**, **Eigenfrequency**, and **Damped eigenfrequency**. The coupled displacement-temperature field presents thermoelastic oscillations of small amplitude, which are initialized to zero. You need to set the strain reference temperature **Tempref** to zero, and use a special form of the heat balance equation. For more details, see the example "Heat Generation in a Vibrating Structure" on page 703 of the *Structural Mechanics Module Model Library*.

Damping

In transient and frequency response analyses you have the possibility to model undamped or damped problems. In the Structural Mechanics Module you can specify damping on the subdomain level using the **Damping** page that appears in the **Subdomain Settings** dialog box. From the **Damping models** list you can select **No damping**, **Rayleigh**, or **Loss factor**, and the contents of the dialog box changes for each of these damping models.



Damping page when Rayleigh damping is selected.

Note: Loss factor damping is valid only for frequency response analysis. If you choose transient analysis and loss factor damping, COMSOL Multiphysics solves the model with no damping.

Table 7-2 and the subsequent text describe the parameters that define damping:

TABLE 7-2: PARAMETERS FOR DAMPING MODELS

PARAMETER	VARIABLE	DESCRIPTION	DAMPING MODEL
α_{dM}	alphadM	Mass-damping parameter	Rayleigh
β_{dK}	betadK	Stiffness-damping parameter	Rayleigh
η	eta	Loss factor	Loss factor

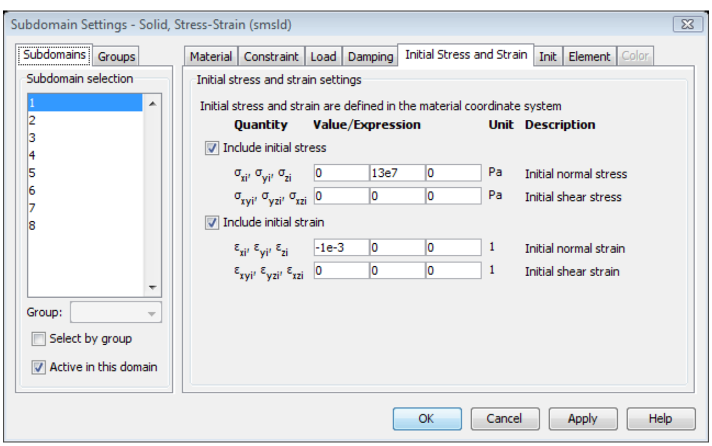
Mass damping parameter Defines the Rayleigh damping model's mass damping, α_{dM}.

Stiffness damping parameter Defines the Rayleigh damping model's stiffness damping, β_{dK}.

Loss factor Defines the loss factor η for the loss factor damping model.

Initial Stress and Strain

An analysis can include initial stress and strain, which is the stress/strain state in the structure before the model applies any constraint or load. Initial strain can, for instance, describe moisture-induced swelling, and initial stress can describe stresses from heating. In fact, you can think of initial stress and strain as different ways to express the same thing. To specify them, go to the **Initial Stress and Strain** page in the **Physics>Subdomain Settings** dialog box.



Dialog box for setting up initial stress and strain.

You control the option to include initial stress and strain independently using like-named two check boxes.

In the following table, the index l for parameter σ_{0l}/`s`l`i` and ε_{0l}/`e`l`i` refers to the space coordinates x_l.

PARAMETER	VARIABLE	DESCRIPTION
σ_{0l}	`s`l`i`	Initial normal stress
τ_{0lk}	`s`lk`i`	Initial shear stress
ε_{0l}	`e`l`i`	Initial normal strain
ε_{0lk}	`e`lk`i`	Initial shear strain

The index l runs over the following coordinate names for the different application modes.

APPLICATION MODE	x_1	x_2	x_3
Plane Stress and Plane Strain	x	y	z
Solid, Stress-Strain	x	y	z
Axial Symmetry, Stress-Strain	r	φ	z

Example: σ_{0l} for axial symmetry stress-strain means σ_{0r}, $\sigma_{0\varphi}$, and σ_{0z}.

The parameters σ_{0lk}/`s`lk`i` and ε_{0lk}/`e`lk`i` in the first table refer to the space coordinates $x_l x_k$ and are defined for the following combinations of l and k for the different application modes:

APPLICATION MODE	$x_1 x_2$	$x_2 x_3$	$x_1 x_3$
Plane Stress and Plane Strain	xy		
Solid, Stress-Strain	xy	yz	xz
Axial Symmetry, Stress-Strain	rz		

Perfectly Matched Layers (PMLs)

In frequency response analysis of elastic waves, you can use perfectly matched layers to simulate absorbing boundaries. A PML is strictly speaking not a boundary condition but an additional domain that absorbs the incident radiation without producing reflections. It provides good performance for a wide range of incidence angles and is not particularly sensitive to the shape of the wave fronts. The PML formulation introduces a complex-valued coordinate transformation under the additional requirement that the wave impedance should not be affected. The following sections describe how to create Cartesian, cylindrical, and spherical PMLs for elastic waves.

For an account of elastic waves in solids, see Chapters 4 and 5 of Ref. 1. For background information about PMLs in elastodynamics, see Ref. 2.

PML IMPLEMENTATION

For a PML that absorbs waves in the coordinate direction ξ, the Structural Mechanics Module uses the following coordinate transformation inside the PML:

$$\xi' = \mathrm{sign}(\xi - \xi_0)|\xi - \xi_0|^n \frac{L}{\delta\xi^n}(1 - i) \tag{7-22}$$

The scaled PML width, L; the coordinate of the inner PML boundary, ξ_0; and the (actual) width of the PML, $\delta\xi$, are input parameters for each orthogonal absorbing coordinate direction.

The scaling exponent, n is an input parameter for each PML subdomain. The default value of n is 1, giving a linear scaling that works well in most cases, and the useful range is roughly between 1 and 2; increasing the exponent allows you to use fewer mesh elements to resolve wavelengths much smaller than the scaled PML width.

Usually, set L equal to one wavelength. The wavelength depends on the type of elastic wave you are considering. For example, for longitudinal (acoustic) waves, the wavelength is given by (Ref. 1)

$$\lambda = \frac{1}{f}\sqrt{\frac{(1-\nu)}{(1+\nu)(1-2\nu)}\frac{E}{\rho}}$$

where f is the frequency, E is Young's modulus, ν is Poisson's ratio and ρ is the density. If your analysis includes several wave types of different wavelengths, set L to the longest one. For this case, you can also try to set the scaling exponent, n, equal to 2.

The parameters ξ_0 and $\delta\xi$ get default settings that the software deduces from the drawn geometry and stores in so-called guess variables. You can inspect the values of the guess variables on the **Variables** page of the **Subdomain Settings - Equation System** dialog box or at the corresponding node of the **Model Tree**.

The default settings defined by the guess variables work nicely in most cases, but they might fail for PML subdomains of nonstandard shape. Examples of geometries that work nicely are shown in the following figures for each of the available PML types:

- **Cartesian**—PMLs absorbing in Cartesian coordinate directions.

- **Cylindrical**—PMLs absorbing in cylindrical coordinate directions from a specified axis. For axisymmetric geometries the cylinder axis is the z-axis.
- **Spherical**—PMLs absorbing in the radial direction from a specified center point.

  For each of the above PML types, you can choose the coordinate directions in which the PML absorbs waves, that is, for which directions a coordinate transformation of the type Equation 7-22 applies. To allow complete flexibility in defining a PML there is, in addition, a fourth option:
- **User defined**—General PMLs or domain scaling with user-defined coordinate transformations.

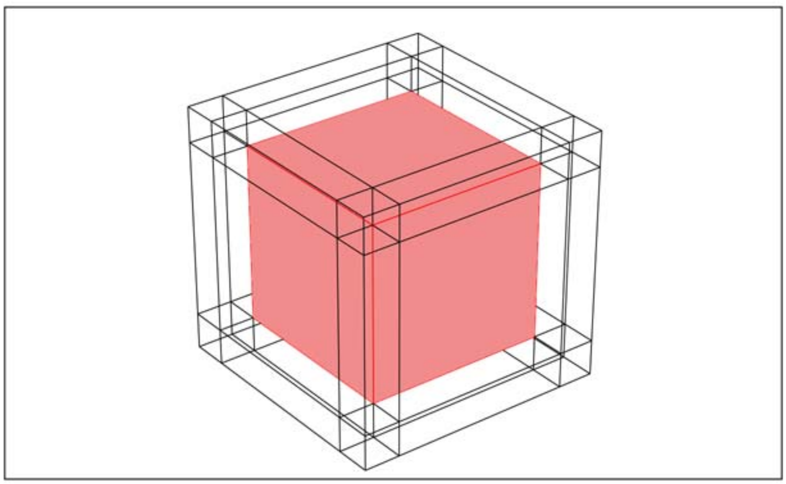


Figure 7-1: A cube surrounded by typical PML regions of the type "Cartesian."

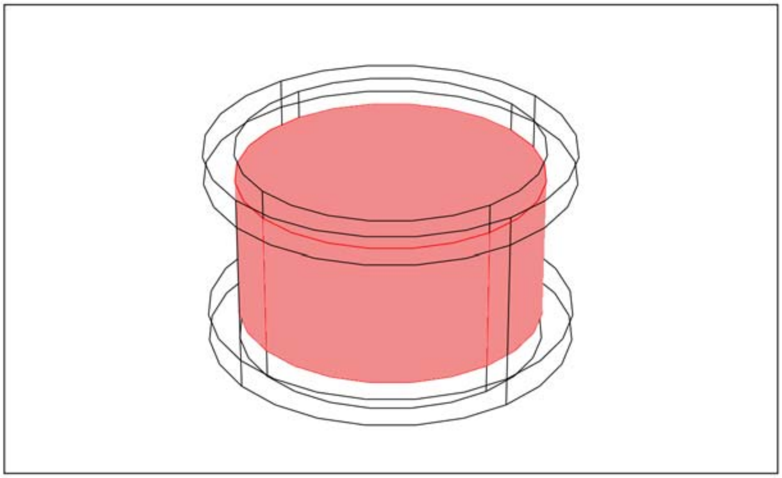


Figure 7-2: A cylinder surrounded by typical cylindrical PML regions.

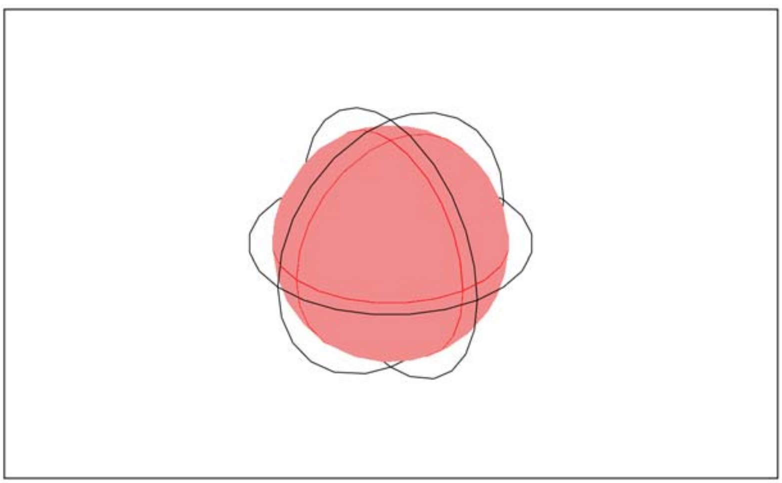


Figure 7-3: A sphere surrounded by a typical spherical PML region.

SETTING UP A PML

To model an absorbing boundary using PMLs, you need an auxiliary subdomain outside the boundary. On the **PML** page in the **Subdomain Settings** dialog box, you can select different types of PMLs depending on what kind of wave you have:

PML TYPE	APPLICATION MODE	DESCRIPTION
None	all	No PML
Cartesian	Solid, Stress-Strain; Plane Stress; and Plane Strain	Absorbs waves in the specified Cartesian coordinate directions
Cylindrical	all	Absorbs cylindrical waves
Spherical	Solid, Stress-Strain; Axial Symmetry, Stress-Strain	Absorbs spherical waves
User defined	all	Define your own scaled space variables

The PML type **None** is default. To add a PML, select one of the other types.

Cartesian PMLs

When choosing the Cartesian PML type you can use a user-defined coordinate system to define the directions. If you want a curved coordinate system you must use the cylindrical or spherical PML type.

Select the check box for the directions in which you want the waves to be absorbed. For each of these directions, enter the scaled PML width, L in the associated edit field. Make sure all material properties are the same in the PML as in the adjacent subdomain.

Cylindrical PMLs

In 2D, a cylindrical PML always absorbs waves in the radial direction. In the other dimensions, you can decide how the PML absorbs the wave: in the radial direction, the z direction, or both.

Select the directions in which you want the PML to absorb the waves and enter the scaled PML widths in those directions. To define a cylindrical PML you also need to enter the center point of the cylindrical coordinates and, in 3D, the cylinder axis direction.

Spherical PMLs

A spherical PML always absorbs waves in the radial direction. Enter the scaled PML width, L. Define the spherical coordinates by entering the center point.

User-Defined PMLs

When using a PML, the algorithm scales the equation in this domain so that instead of the coordinates used in the rest of the model, the coordinates PMLx, PMLy, and PMLz appear in the equation. If you want to scale the equation in some other way than the automatic PML options provide, use a user-defined PML. In this case you enter your own **User-defined PML coordinates**.

REFERENCES

1. L.M. Brekhovskikh and V. Goncharov, *Mechanics of Continua and Wave Dynamics*, 2nd ed., Springer-Verlag, 1994.

2. W.C. Chew and Q.H. Liu, "Perfectly Matched Layers for Elastodynamics: A New Absorbing Boundary Condition," *J. Comp. Acoustics*, vol. 4, pp. 341–359, 1996.

Contact Modeling

You can model contact between two boundaries. The boundaries need to be connected to a subdomain active in the same application mode. To be able to model contact you need the following:

- An application mode modeling the deformation that supports contact modeling: The Plane Strain; Plane Stress; Axial Symmetry, Stress-Strain; or the Solid Stress-Strain application mode.
- A deformed frame controlled by the application mode. This is done by setting the application mode property **Create frame** to **On**. The program does this automatically when you add a contact pair.
- Use of assembly mode, if the parts are in contact initially. In this case select **Use Assembly** from the **Draw** menu. Read more about assemblies in the section "Using Assemblies" on page 351 in the *COMSOL Multiphysics Modeling Guide*.
- A contact pair. A contact pair consists of a number of slave and master boundaries. The slave is constrained not to penetrate the master boundary. You can create contact pairs from the **Contact Pairs** dialog box, which you open from the **Physics** menu. A description of how to create contact pairs appears later in this section. Additionally, if some parts of the boundaries are in initial contact, you can use the **Create Pairs** dialog box to automatically detect and define contact pairs. For more information about the **Create Pairs** dialog box, see the section "Creating Pairs" on page 353 in the *COMSOL Multiphysics Modeling Guide*.
- Contact parameters, specified to suit your model. You can inspect and change the contact parameters in the **Boundary Settings** dialog box by selecting the appropriate

contact pair on the **Pair** page. There are three different pages, described further on in this section, to do this on:

  - **Contact**
  - **Contact, Initial**
  - **Contact, Advanced**

- Solver parameters, specified to suit your model. You can set the parameters for the augmented and nonlinear solvers in the **Solver Settings** dialog box, which is described in the section "Nonlinear Solver Settings" on page 368 in the *COMSOL Multiphysics User's Guide*. You can find recommendations for solver settings specifically for contact models on page 129 of this book.

Note: The current version supports contact in the continuum application modes: Plane Stress; Plane Strain; Axial Symmetry, Stress-Strain; and Solid, Stress-Strain.

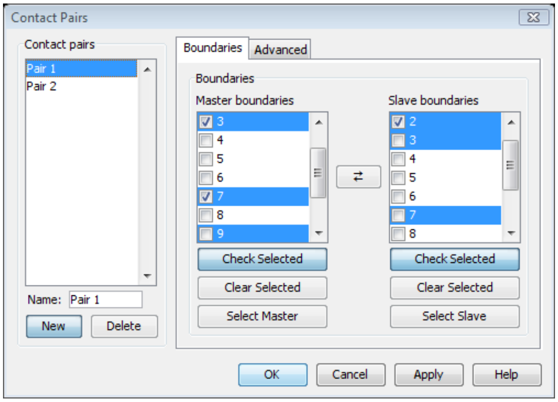
Note: You cannot use contact pairs together with the transient solver. Use the parametric solver with the time *t* as the parameter to solve contact problems with friction.

This section provides information about how to set up contact pairs and specify contact parameters. You can read about the theory behind the implementation of the contact modeling method on page 186 and about tips for creating a contact model on page 118.

For general information about modeling with pairs, see the section "Specifying Physics Settings on Pairs" on page 361 in the *COMSOL Multiphysics Modeling Guide*.

THE CONTACT PAIRS DIALOG BOX

To define contact pairs, choose **Physics>Contact Pairs**. This opens the **Contact Pairs** dialog box.



Boundaries Page

Each pair has a name. The application modes use this name to refer to the pair. The name must be unique.

The two boundary lists show the master and slave domains of the pair selected in the list to the left. The check boxes beside the domain numbers indicate which domains belong to the master and the slave, respectively.

Clicking the **Check Selected** buttons below the lists selects the check boxes of the boundaries highlighted in the list. This is equivalent to selecting the individual check boxes and is a quick way to select multiple check boxes. Clicking the **Clear Selected** button similarly clears the check boxes of the selected domains.

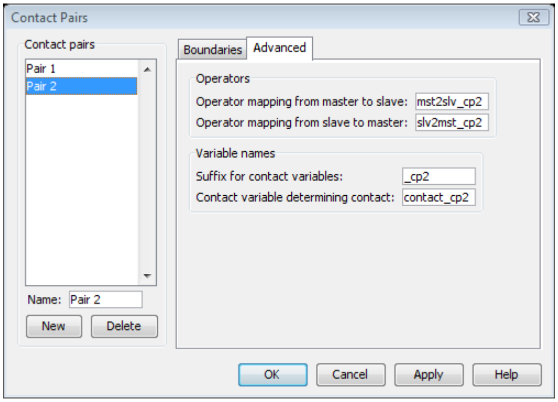
Use the buttons **Select Master** and **Select Slave** to select the master and slave domains in the main window and in the selection lists.

Clicking the arrow button between the selection lists interchanges the master and slave domains.

For best results you should consider the guidelines on page 118 when selecting your master and slave boundaries.

Advanced Page

On the **Advanced** page in the **Contact Pairs** dialog box you can define the names of the contact pair's coupling operators.



When creating a contact pair COMSOL Multiphysics automatically defines the operators and gives them a name. The names have to be unique within the whole model. The application modes use the operators to set up the contact condition preventing the slave from penetrating the master.

A *map operator* evaluates its argument on one side of the pair and makes the result available on the other side. In the previous figure you can see two operators: `mst2slv_cp2`, mapping from the master of the pair to the slave, and `slv2mst_cp2`, mapping in the other direction. For example, if `u` is a variable on the master side you can use the expression `mst2dst_cp2(u)` on the slave side.

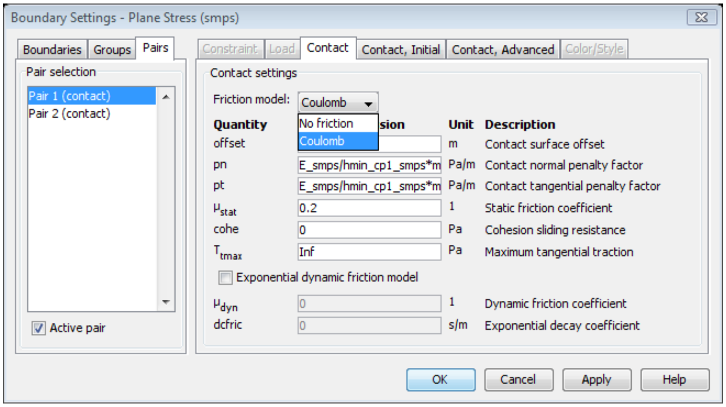
The application modes create a number of variables on the boundaries of the contact pair. To make the variable names unique the software adds a suffix to them. You can edit the suffix name in the **Suffix for contact variables** edit field. A special variable is the contact variable stating if the boundaries are in contact or not, which you can use in logical expressions. The expression `if(contact_cp1,1,2)` on the slave side evaluates to 1 for points where the boundaries are in contact and to 2 in the other points. You can edit the name of the contact variable in the **Contact variable determining contact** edit field.

BOUNDARY SETTINGS DIALOG BOX

The **Boundary Settings** dialog box, which you can open from the **Physics** menu, has three pages dedicated for contact settings: the **Contact**; **Contact, Initial**; and **Contact, Advanced** pages. In the following you can find a description of these.

Contact Page

You specify the most important settings for your contact problem on the **Contact** page.



The Contact page for the Plane Stress application mode.

The **Active pair** check box lets you select if you want to use the contact pair in this application mode or not. If you want to model friction between the contact pairs, select **Coulomb** from the **Friction model** list.

The following table specifies the contact pair parameters on the **Contact** page:

PARAMETER	DESCRIPTION	SI UNIT	NO FRICTION	COULOMB
offset	An optional offset specifying at what distance from the geometrical boundary contact appears, positive in the normal direction	m	√	√
pn	The normal penalty factor	Pa/m	√	√
pt	The tangential penalty factor	Pa/m		√
μ_{stat}	Static friction coefficient	-		√
cohe	Cohesion sliding resistance, the friction force at zero contact pressure	Pa		√
T_{tmax}	The maximum tangential traction	Pa		√

PARAMETER	DESCRIPTION	SI UNIT	NO FRICTION	COULOMB
μ_{dyn}	Dynamic friction coefficient, only used with the dynamic friction option	-		√
dcfric	Decay coefficient, only used with the dynamic friction option	s/m		√

The convergence is sensitive to the value of the penalty factors. Their value should be of the same order as the stiffness of the boundary divided by a typical length scale, that is, the mesh size. The default value for both the normal and tangential penalty factors is set according to

$$\mathrm{p} = \frac{E}{h_{\mathrm{min}}} \cdot \min(10^{-3} \cdot 5^{\mathrm{auglagiter}}, 1) \tag{7-23}$$

The Young's modulus is denoted E and the smallest mesh size on the slave boundary, h_{min}, is included in order to get a typical length scale. The `auglagiter` variable is the iteration number in the augmented Lagrange solver. It is used to make the penalty parameter soft at the beginning (to help the solver get started) and to gradually make it stiffer (to speed up convergence).

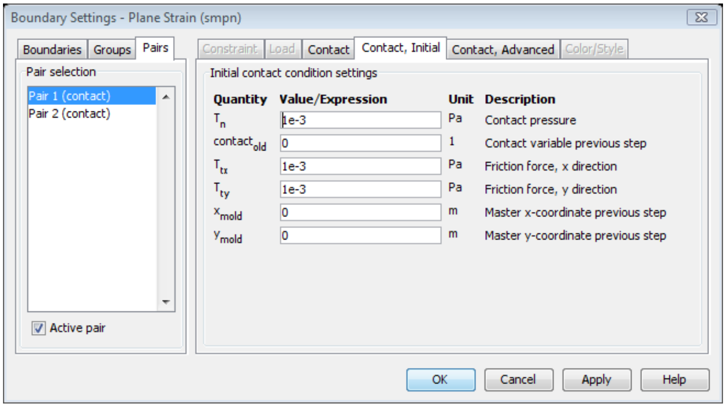
The default values, using Young's modulus only work for linear isotropic materials, for which the Young's modulus is defined. For other types of materials you need to substitute E with a suitable value or define it as a constant or expression variable. Read more about selecting the penalty factor on page 119.

If you select the **Exponential dynamic friction model** check box you get a friction coefficient that varies between the static and dynamic friction coefficient depending on the slip velocity and the dcfric decay coefficient in the following way.

$$\mu_{\mathrm{dyn}} + (\mu_{\mathrm{stat}} - \mu_{\mathrm{dyn}})e^{-\mathrm{dcfric}|v_s|} \tag{7-24}$$

Contact, Initial Page

You specify the initial conditions for your contact problem on the **Contact, Initial** page.



The Contact, Initial page for the Plane Strain application mode.

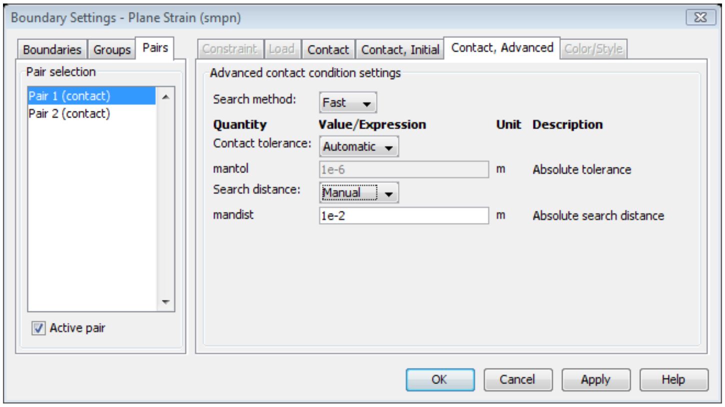
The following table specifies the contact pair parameters on the **Contact, Initial** page:

PARAMETER	DESCRIPTION	SI UNIT	NO FRICTION	COULOMB
T_n	The initial value for the contact pressure.	Pa	√	√
$contact_{old}$	The initial value for the `contact` variable in the previous step.			√
T_{txi}	The initial value for the friction force components.	Pa		√
xi_{mold}	The initial value for the coordinates of the master point in the previous step.	m		√

Turn to page 120 to read about how the initial contact pressure can influence your contact model.

Contact, Advanced Page

You have the option to specify more advanced contact pair settings on the **Contact, Advanced** page.



The Contact, Advanced page for the Plane Strain application mode.

You specify what search method to use in the **Search method** list. The default option is **Fast**. Under some rare circumstances this method can fail to detect contact and find the corresponding master points. Then select the more robust but slower option **Direct** instead.

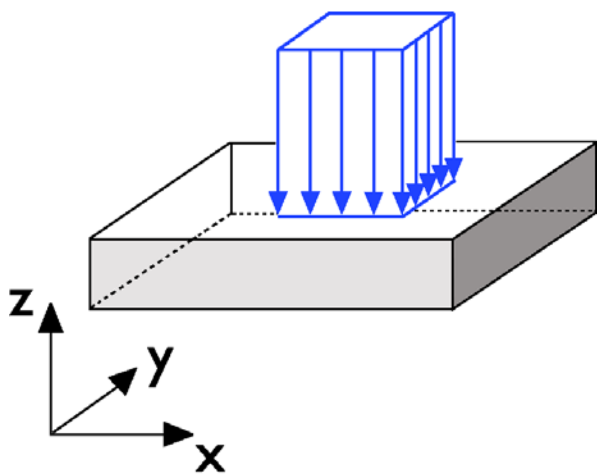
You have two options to calculate the contact tolerance: **Automatic** or **Manual**. That is, at what distance between the two bodies they are regarded as being in contact (used for friction and multiphysics contact). You select this from the **Contact tolerance** list. **Automatic** means that the software calculates the tolerance from the size of the bounding box of the total geometry. **Manual** means that you specify the value yourself in the **mantol** edit field.

In a similar way as for the contact tolerance you have the option to specify the **Search distance**. The search distance sets the radius from any slave point within which the program looks for possible contact between the slave and master boundary. A shorter distance speeds up the search algorithm because the vast majority of boundary elements can quickly be excluded from the search process. But a too small value might result in missed contact detection.

The **Automatic** setting means that the program calculates the search distance from the size of the bounding box of the total geometry. If the total size of the geometry is not representative for the size of the contact areas, you can use the **Manual** setting and specify the value yourself in the **mandist** edit field. A suitable search radius is usually on the order of the largest mesh elements involved in the contact process.

8

Mindlin Plates



A plate is a thin planar structure, its thickness as a rule being less than one tenth of its width. In contrast to the plane stress and plane strain 2D cases, the forces are either applied in the direction normal to the plate, or as moments about directions in the plane where the plate lies. The main deformation takes place in the out-of-plane direction. There are two main groups of plates:

- Thin plates
- Thick plates

In thin plate theory the transverse shear deformation is neglected, in the same way as Euler beams neglect shear deformations.

In thick plate theory the transverse shear deformation is included. The Mindlin plate is based on the following engineering assumption: a plane originally perpendicular to the mid surface remains plane after loading, but not necessarily perpendicular to the deformed mid surface. The change in angle accounts for the transverse shear deformation.

The element in this application mode is a discrete Reissner-Mindlin triangle. This element has six nodes and a total of twelve degrees of freedom. These are the two rotations and one transversal displacement at each corner node and the normal rotations at the triangle midsides; see O. C. Zienkiewicz (Ref. 1) for details.

Variables and Space Dimensions

The dependent variables are the global displacement w in the z direction and the rotations θ_x and θ_y around the global x- and y-axes.

Reference

1. O. C. Zienkiewicz: "*Plate bending elements with discrete constraints: new triangular elements*," Computer & Structures, vol. 35, no. 4, 1990.

Theory Background

Shape Function

The degrees of freedom are defined by a shape function object `shdrm`. The degree of freedom names and variable names are constructed from the input arguments.

```
shdrm('w','thx','thy')
```

The `shdrm` shape function object defines the following dependent variables, derivatives of dependent variables and shear strain components.

VARIABLE	NAME	DOF	DESCRIPTION
w	`w`	`w`	Global displacement in z direction
θ_x	`thx`	`thx`	Rotation about global x-axis
θ_y	`thy`	`thy`	Rotation about global y-axis
	`thn`	`thn`	Midside rotation about axis perpendicular to side (with a direction convention)
$\frac{\partial\theta_x}{\partial x}$	`thxx`		x derivative of rotation about global x-axis
$\frac{\partial\theta_x}{\partial y}$	`thxy`		y derivative of rotation about global x-axis
$\frac{\partial\theta_y}{\partial y}$	`thyy`		y derivative of rotation about global y-axis
$\frac{\partial\theta_y}{\partial x}$	`thyx`		x derivative of rotation about global y-axis
γ_{xz}	`gxz`		Shear strain component
γ_{yz}	`gyz`		Shear strain component

The shape functions are of order 1 for the out-of-plane displacements, partly order 2 for rotations (rotations about triangle sides vary linearly), and partly order 1 for shears (shear components along triangle sides are constant). See `shdrm` on page 160 in the *Structural Mechanics Module Reference Guide* for details.

In-Plane Strain-Displacement/Rotation Relation

The in-plane strain components depends on the rotation derivatives defined by the `shdrm` shape function and the z coordinate in the plate.

$$\varepsilon = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} = z \begin{bmatrix} \dfrac{\partial\theta_y}{\partial x} \\ -\dfrac{\partial\theta_x}{\partial y} \\ \left(\dfrac{\partial\theta_y}{\partial y} - \dfrac{\partial\theta_x}{\partial x}\right) \end{bmatrix} = z\Theta$$

The total strain ε consists of thermal ($\varepsilon_{\mathrm{th}}$), initial ($\varepsilon_{\mathrm{i}}$), and elastic strains($\varepsilon_{\mathrm{el}}$)

$$\varepsilon = \varepsilon_{\mathrm{el}} + \varepsilon_{\mathrm{th}} + \varepsilon_{\mathrm{i}}$$

Transverse Strain Components

The average transverse shear components is defined directly by the `shdrm` shape function.

In-Plane Stress-Strain Relation

The in-plane stress components in the plate are described by the symmetric stress tensor

$$\sigma = \begin{bmatrix} \sigma_x & \tau_{xy} \\ \tau_{yx} & \sigma_y \end{bmatrix} \qquad \tau_{xy} = \tau_{yx}$$

consisting of 2 normal stresses (σ_x and σ_y) and two or, if the symmetry is used, one shear stress τ_{xy}. The stress-strain relation for linear conditions including initial stress and strain and thermal effects reads:

$$\sigma = D_p\varepsilon_{el} + \sigma_i = D_p(\varepsilon - \varepsilon_{th} - \varepsilon_i) + \sigma_i \qquad (8\text{-}1)$$

where D_p is the 3-by-3 elasticity matrix in plane stress form. The stress and strain components are described on vector form with the three stress and strain components in column vectors defined as

$$\sigma = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} \qquad \varepsilon = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix}$$

Note: In the following descriptions the compact notation σ and ε will be used meaning either the stress/strain vector or tensor depending on the situation.

The in-plane elasticity matrix D_p and the more basic matrix, the flexibility or compliance matrix D_p^{-1} (the inverse of D_p), are defined differently for isotropic, orthotropic, and anisotropic materials. For isotropic material the D_p^{-1} matrix looks like

$$D_p^{-1} = \frac{1}{E}\begin{bmatrix} 1 & -\nu & 0 \\ -\nu & 1 & 0 \\ 0 & 0 & 2(1+\nu) \end{bmatrix}$$

where E is the modulus of elasticity or *Young's modulus* and ν is *Poisson's ratio*, defining the contraction in the perpendicular direction. Inverting D_p^{-1} symbolically results in

$$D_p = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

where E is Young's modulus and ν is Poisson's ratio. For an orthotropic material the D_p^{-1} matrix looks like.

$$D_p^{-1} = \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{yx}}{E_y} & 0 \\ -\frac{\nu_{xy}}{E_x} & \frac{1}{E_y} & 0 \\ 0 & 0 & \frac{1}{G_{xy}} \end{bmatrix}$$

The D_p^{-1} matrix is symmetric so the material is defined using the coefficients on the lower-diagonal part of the matrix. It is important that the material data has been derived using the same definition of ν as above; if not, the material data need to be transformed.

Note: ν_{ij} is defined in different ways depending on the application field. It is easy to transform between the different definitions but you need to check what definition your material uses.

Inverting the D_p^{-1} matrix symbolically using only the E_x, E_y, ν_{xy}, and G_{xy} coefficients results in the following symmetric D_p matrix.

$$D_p = \begin{bmatrix} Dp_{11} & Dp_{12} & 0 \\ Dp_{12} & Dp_{22} & 0 \\ 0 & 0 & Dp_{33} \end{bmatrix}$$

where the components are as follows

$$Dp_{11} = \frac{E_x^2}{D_{denom}} \qquad Dp_{12} = \frac{E_x E_y \nu_{xy}}{D_{denom}}$$

$$Dp_{22} = \frac{E_x E_y}{D_{denom}} \qquad Dp_{33} = G_{xy}$$

where

$$D_{denom} = E_x - E_y \nu_{xy}^2$$

For an anisotropic material the symmetric D_p matrix is given explicitly.

Note: For an anisotropic material the D_p matrix should be given in plane stress form, using the equation $\sigma_z = 0$ to eliminate ε_z. If material data is given in full 3D form they need to be transformed to plane stress form using the $\sigma_z=0$ condition.

Transverse Stress-Strain Relation

The average shear strain γ_m is defined as

$$\gamma_m = \frac{Q}{G \cdot \frac{\mathrm{th}}{S_\mathrm{f}}}$$

where:

- G is the shear modulus
- Q is the plate shear force/length
- th is the thickness of the plate
- S_f is the shear factor

The shear factor is defined so that the average strain and the real strain should result in equal virtual work through the thickness.

$$\int_{\mathrm{th}} \gamma\tau dz = \mathrm{th} \cdot Q\gamma_m$$

Assuming a parabolic stress and strain distribution through the plate results in $S_f = 1.2$, this holds for homogeneous plates.

For the general case the relation looks like

$$\begin{bmatrix} Q_y \\ Q_x \end{bmatrix} = \mathrm{th} \cdot D_s \begin{bmatrix} \gamma_{yzm} \\ \gamma_{xzm} \end{bmatrix}$$

Where D_s looks as follows for the different material models:

- Isotropic

$$D_s = \begin{bmatrix} \frac{E}{2(1+\nu)S_\mathrm{f}} & 0 \\ 0 & \frac{E}{2(1+\nu)S_\mathrm{f}} \end{bmatrix}$$

- Orthotropic

$$D_s = \begin{bmatrix} \frac{G_{yz}}{S_{\mathrm{f}yz}} & 0 \\ 0 & \frac{G_{xz}}{S_{\mathrm{f}xz}} \end{bmatrix}$$

- Anisotropic: The full elasticity matrix including shear factors D_s is given explicitly.

Thermal Strain

The thermal strain is only included for the in-plane strain components. The temperature is assumed to vary linearly through the thickness.

$$T = T_0 + \Delta T \frac{z}{\mathrm{th}}$$

The plate can only handle the temperature difference through the plate, ΔT. The thermal strain as a function of the z-coordinate and the temperature gradient is

$$\varepsilon_{\mathrm{th}} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix}_{\mathrm{th}} = \alpha_{\mathrm{vec}} \Delta T \frac{z}{\mathrm{th}} \qquad (8\text{-}2)$$

Depending on the material model, α_{vec} is set up differently:

- Isotropic

$$\alpha_{\mathrm{vec}} = \begin{bmatrix} \alpha \\ \alpha \\ 0 \end{bmatrix}$$

- Orthotropic

$$\alpha_{\mathrm{vec}} = \begin{bmatrix} \alpha_x \\ \alpha_y \\ 0 \end{bmatrix}$$

- Anisotropic: The full thermal expansion vector is given as input.

Initial Load and Strain

IN-PLANE MOMENTS AND CURVATURE

The initial stress means the stress before any loads, displacements, and initial strains have been applied.

$$\sigma_i = \begin{bmatrix} \sigma_{xi} \\ \sigma_{yi} \\ \tau_{xyi} \end{bmatrix}$$

The initial stress distribution is given as initial plate moments.

$$M_{xpi} = \int_{th} \sigma_{xi} z dz \qquad M_{ypi} = \int_{th} \sigma_{yi} z dz \qquad M_{xypi} = \int_{th} \tau_{xyi} z dz$$

The initial strain is the strain before any loads, displacements, and initial stresses have been applied.

$$\varepsilon_i = \begin{bmatrix} \varepsilon_{xi} \\ \varepsilon_{yi} \\ 2\varepsilon_{xyi} \end{bmatrix} = z\Theta_i$$

The initial strain distribution is given as initial curvature and warping.

$$\varepsilon_{xi} = z\theta_{yxi} \qquad \varepsilon_{xi} = -z\theta_{xyi} \qquad \varepsilon_{xyi} = \frac{z}{2}(\theta_{yyi} - \theta_{xxi}) = \frac{z}{2}\theta_{yymxxi}$$

INITIAL SHEAR FORCES AND TRANSVERSAL STRAINS

The out-of-plane initial shear stress is given as shear forces.

$$Q_{yi} = \int_{\mathrm{th}} \tau_{yzi} dz \qquad Q_{xi} = \int_{\mathrm{th}} \tau_{xzi} dz$$

The initial shear strain is given as average shear strains γ_{yzi} and γ_{xzi}.

Implementation

The implementation is based on the principle of virtual work. The principle of virtual work states that the virtual work from any variation in internal strain and external loads are zero. The in-plane part of the virtual work is expressed using the internal plate moments

$$M_p = \begin{bmatrix} M_{xp} \\ M_{yp} \\ M_{xyp} \end{bmatrix} = \int_{\mathrm{th}} z[D_p(\varepsilon - \varepsilon_{\mathrm{th}} - \varepsilon_i) + \sigma_i]dz =$$

$$\int_{\mathrm{th}} z^2 \left[D_p\left(\Theta - \frac{\alpha_{\mathrm{vec}}\Delta T}{\mathrm{th}} - \Theta_i\right)\right]dz + M_{pi} =$$

$$\frac{(\mathrm{th})^3}{12}\left[D_p\left(\Theta - \frac{\alpha_{\mathrm{vec}}\Delta T}{\mathrm{th}} - \Theta_i\right)\right] + M_{pi}$$

The out-of-plane part is expressed using the internal shear forces:

$$Q_p = \begin{bmatrix} Q_{yp} \\ Q_{xp} \end{bmatrix} = \int_{\mathrm{th}} z\tau dz = \mathrm{th} \cdot D_s\left[\begin{bmatrix} \gamma_{yzm} \\ \gamma_{xzm} \end{bmatrix} - \begin{bmatrix} \gamma_{yzi} \\ \gamma_{xzi} \end{bmatrix}\right] + Q_{pi}$$

The variation of the total stored energy W from external and internal strain and load is

$$\delta W = -\int_A \left(\left(\frac{\partial \theta_y}{\partial x}\right)_{\mathrm{test}} M_{xp} - \left(\frac{\partial \theta_x}{\partial y}\right)_{\mathrm{test}} M_{yp} + \left(\frac{\partial \theta_y}{\partial y} - \frac{\partial \theta_x}{\partial x}\right)_{\mathrm{test}} M_{xyp}\right.$$
$$\left. + 2Q_{yp}\gamma_{yz\mathrm{test}} + 2Q_{xp}\gamma_{xz\mathrm{test}} + w_{\mathrm{test}}F_{zg} + \theta_{x\mathrm{test}}M_{xg} + \theta_{y\mathrm{test}}M_{yg}\right)dA$$

If the material is described in a local user-defined coordinate system, the variational equation is expressed in local instead of global plate moments and shear forces.

The rotational derivatives can be transformed as a tensor.

$$\begin{bmatrix} \theta_{yx} & \frac{1}{2}(\theta_{yy} - \theta_{xx}) \\ \frac{1}{2}(\theta_{yy} - \theta_{xx}) & -\theta_{xy} \end{bmatrix}_l = T_{\mathrm{coord}}{}^T \begin{bmatrix} \theta_{yx} & \frac{1}{2}(\theta_{yy} - \theta_{xx}) \\ \frac{1}{2}(\theta_{yy} - \theta_{xx}) & -\theta_{xy} \end{bmatrix} T_{\mathrm{coord}}$$

where T_{coord} is the local to global coordinate system transformation matrix.

The local plate moments are then calculated from the local rotational derivatives.

The global plate moments are calculated by transforming the local plate moments.

$$M_p = T_{\text{coord}} M_{p1} T_{\text{coord}}^{\;T}$$

The shear strains transforms as

$$\begin{bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{bmatrix}_1 = T_{\text{coord}}^{\;T} \begin{bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{bmatrix}$$

The global shear forces are calculated by transforming the local shear forces.

$$\begin{bmatrix} Q_{xp} \\ Q_{yp} \end{bmatrix} = T_{\text{coord}} \begin{bmatrix} Q_{xp} \\ Q_{yp} \end{bmatrix}_1$$

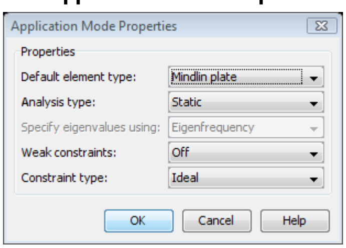
Application Mode Description

This section describes how to define a Mindlin Plate model. It is divided into the following sections:

- Properties
- Scalar Variables
- Material
- Constraint
- Load
- Thermal Coupling
- Initial Stress and Strain
- Postprocessing

Properties

The **Application Mode Properties** dialog box is opened from the **Physics** menu.



In the **Application Mode Properties** dialog box you control different global settings for the model.

- **Analysis type**: A list of different analyses to perform. It affects both the equations and what solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. The available analysis types use the following solvers.

ANALYSIS TYPE	COMSOL MULTIPHYSICS SOLVER
Static	Stationary
Eigenfrequency	Eigenvalue
Time dependent (Transient)	Time dependent
Frequency response	Parametric

ANALYSIS TYPE	COMSOL MULTIPHYSICS SOLVER
Parametric	Parametric
Quasi-static transient	Time dependent

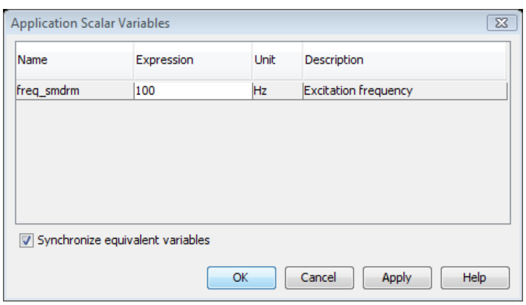
- **Weak constraints**: Controls whether or not weak constraints are active. Use weak constraints for accurate reaction-force computation. When weak constraints are enabled, all constraints are weak by default, but it is possible to change this setting for individual domains.
- **Constraint type**: Constraints can be ideal or nonideal (see “Ideal vs. Non-Ideal Constraints” on page 301 in the *COMSOL Multiphysics Modeling Guide*).

Scalar Variables

There are two different scalar variables:

- Excitation frequency, `freq`, applicable only for frequency response analysis.
- Complex angular frequency, `jomega`, applicable only for eigenfrequency analysis. You normally do not need to edit the complex angular frequency.

The **Scalar Variables** menu item on the **Physics** menu is enabled only when **Frequency Response**, **Damped Eigenfrequency**, or **Eigenfrequency** is selected as **Analysis type** in the **Application Mode Properties** dialog box.

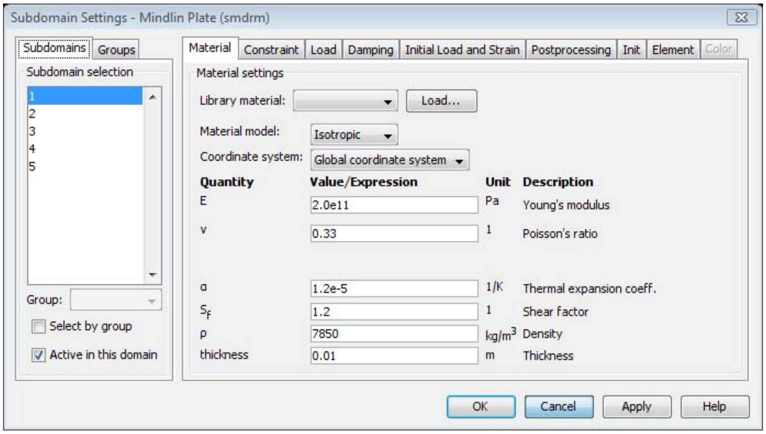


The excitation frequency is the frequency of the harmonic loads in a frequency response analysis.

When frequency response is selected as analysis type, the default solver is the parametric solver making it easy to perform a frequency sweep over several excitation frequencies in a single analysis. In this case `freq_smdrm` is entered as the **Parameter name** on the **Parametric** page in the **Solver Parameters** dialog box and the values entered in the **Parameter values** edit field override the excitation frequency entered in the **Application Scalar Variables** dialog box.

Material

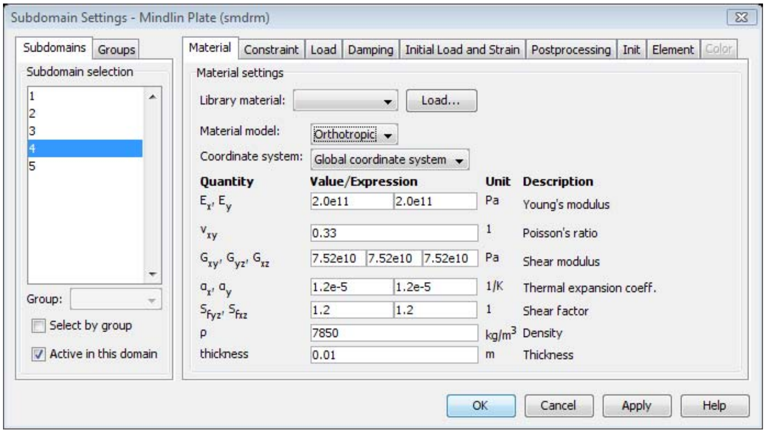
The material properties are defined on the **Material** page in the **Subdomain Settings** dialog box.



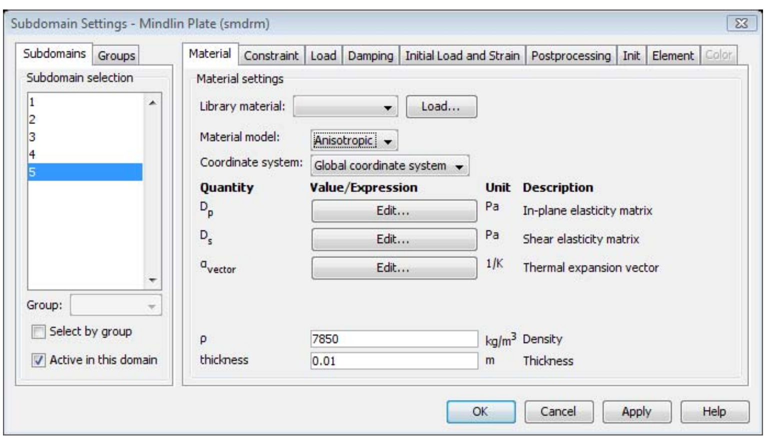
The **Material** page has two lists:

- **Material model**: Select the type of material. Depending on the selection, different material properties are shown reflecting the chosen material model.
  - **Isotropic**: The material has the same material properties in all directions. The **Material** page for an isotropic material is shown above.
  - **Orthotropic**: The material has different material properties in different directions. The in-plane stiffness is defined from the material properties E_x, E_y, ν_{xy}, and G_{xy}; see page 230 for details. The out-of-plane shear stiffness is defined from the properties G_{yz}, G_{xz}, S_{fyz}, S_{fxz}; see page 233 for details. The thermal expansion is

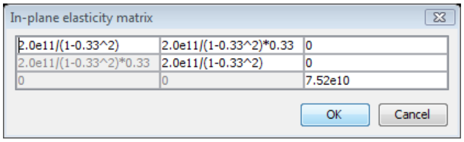
defined from the α_x and α_y; see page 234 for details.The **Material** page for an orthotropic material is shown below.

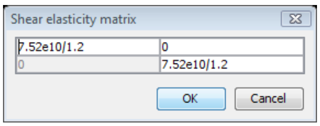


- **Anisotropic**: The material has different material properties in different directions, and the stiffness is defined from the symmetric *elasticity matrices* D_p and D_s; see page 230 and page 233 for details. The thermal expansion is defined from the *thermal expansion vector* α_{vec}; see page 234 for details. The **Material** page for an anisotropic material is shown below.

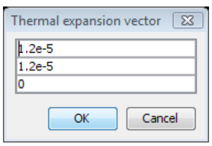


- The **Elasticity matrix** dialog boxes for entering of the D_p and D_s matrices components are shown below.





- The **Thermal expansion vector** dialog box for entering of the α_{vector} is shown below.



- **Coordinate system**: Select the coordinate system where the material properties are defined. This is used for orthotropic and anisotropic materials defined in another coordinate system than the global or if postprocessing variables are needed in a local coordinate system. The **Coordinate system** list is disabled if no user-defined coordinate systems are available. The **Coordinate System Settings** dialog box is found on the **Options** menu. Read more about creation of coordinate systems and their use in "Coordinate Systems" on page 144.

The material properties for the union of all different material models are shown in the table below.

PARAMETER	VARIABLE	DESCRIPTION	MATERIAL MODEL
E	E	Young's modulus	Isotropic
ν	nu	Poisson's ratio	Isotropic
S_{f}	Sf	Shear factor	Isotropic
ρ	rho	Density	All
th	thickness	Thickness	All
α	alpha	Thermal expansion coefficient	Isotropic
E_x, E_y	Ex, Ey	Young's modulus in the x and y directions	Orthotropic
ν_{xy}	nuxy	Shear modulus for the xy-plane	Orthotropic

PARAMETER	VARIABLE	DESCRIPTION	MATERIAL MODEL
G_{xy}, G_{yz}, G_{xz}	`Gxy, Gyz, Gxz`	Poisson's ratio for the xy-, yz-, and xz-planes	Orthotropic
$S_{\mathrm{f}yz}$, $S_{\mathrm{f}xz}$	`Sfyz, Sfxz`	Shear factor for the yz- and xz-planes	Orthotropic
α_x, α_y	`alphax, alphay`	Thermal expansion coefficient in the x and y directions	Orthotropic
D_p		In-plane elasticity matrix for the anisotropic case	Anisotropic
D_s		Out-of-plane elasticity matrix for the anisotropic case	Anisotropic
α_{vec}		Thermal expansion vector for the anisotropic case	Anisotropic

Young's modulus Defines the modulus of elasticity, E, of the material. For an isotropic material, it is the spring stiffness in Hooke's law, shown below in 1D form

$$\sigma = E\varepsilon$$

where σ is the stress and ε is the strain. Orthotropic material uses one value of Young's modulus for each direction, E_i defined on page 230.

Poisson's ratio Denoted by ν, Poisson's ration defines the normal strain in the perpendicular direction, generated from a normal strain in the other direction.

$$\varepsilon_{\perp} = -\nu\varepsilon_{||}$$

For orthotropic material ν_{xy} is defined, see page 230 for details.

Note: ν_{ij} is defined in different ways depending on the application field (see page 230 for details on the definition in the Structural Mechanics Module). It is easy to transform between the different definitions, but you need to check what definition your material uses.

Shear Factor Denoted by S_f, affects the out-of-plane shear stiffness, for homogeneous material $S_f = 1.2$.

Shear Modulus Denoted by G_{ij}, defines the relation between engineering shear strain and shear stress, it is only used for orthotropic material.

$$\gamma_{ij} = \frac{\tau_{ij}}{G_{ij}}$$

Density This material property, ρ, specifies the density of the material.

Thickness Defines the thickness of the plate.

Thermal expansion coefficient Defines how much a material expands due to an increase in temperature.

$$\varepsilon_{\mathrm{th}} = \alpha\Delta T\frac{z}{\mathrm{th}}$$

where $\varepsilon_{\mathrm{th}}$ is the thermal strain, ΔT is the temperature difference through the plate, and α is the thermal expansion coefficient. It is used to model thermal strain for an isotropic material. For an orthotropic material two different thermal expansion coefficients, α_i, are defined for the two perpendicular directions.

Shear Factor Orthotropic Material Denoted by $S_{\mathrm{f}yz}$ and $S_{\mathrm{f}xz}$, these individual shear factors for orthotropic materials affect the out-of-plane shear stiffness.

In-plane elasticity matrix Defines the in-plane elasticity matrix D_p, used for anisotropic materials. See page 230 for details.

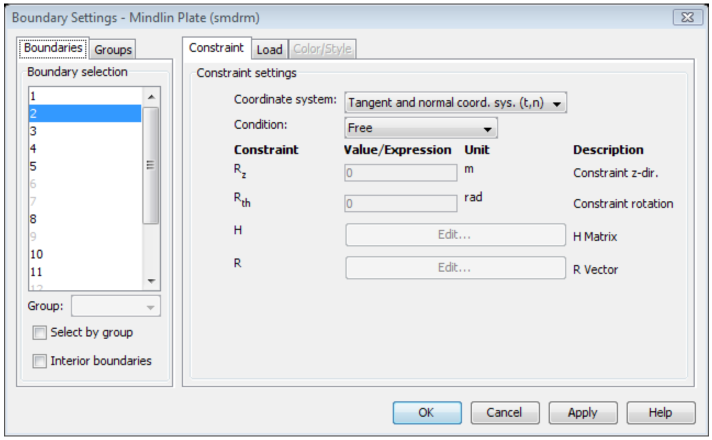
Out-of-plane elasticity matrix Defines the out-of-plane elasticity matrix D_s, used for anisotropic materials. See page 233 for details.

Thermal expansion vector Defines the thermal expansion vector α_{vec}, used for anisotropic materials. See page 234 for details.

Constraint

A constraint specifies the out-of plane displacement and rotations of certain parts of a plate. Constraints can be defined on all domain levels such as points, boundaries, and subdomains. The constraint is controlled from the **Constraint** page in the **Subdomain Settings**, **Boundary Settings**, and **Point Settings** dialog boxes. Normally, you only apply constraints to boundaries.

Below is the **Boundary Settings** dialog box.



With the **Coordinate system** list you control in what coordinate system the constraint is defined. Available options are:

- Tangential and normal coordinate system.
- Global coordinate system.
- User-defined coordinate systems, if there are any local coordinate systems defined. Read more about creation of coordinate system in "Coordinate Systems" on page 144.

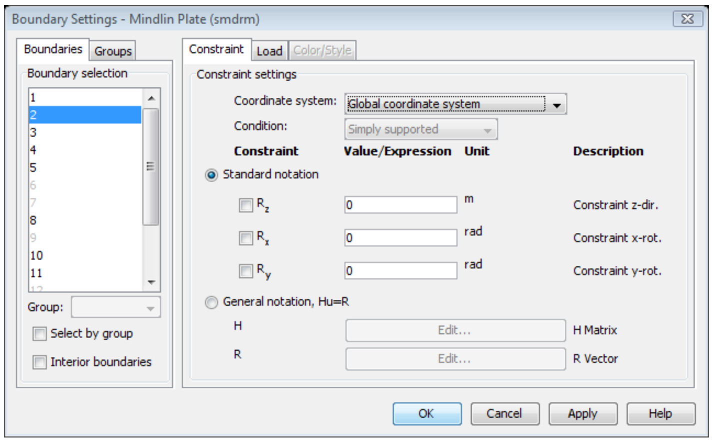
Using the tangential and normal coordinate system you select the constraint condition on the boundary. Available conditions are:

- Free
- Simply supported—the displacement is constrained and the normal rotation is set to zero.
- Fixed—the displacement and tangential rotation is constrained.
- Rotation constrained—the tangential rotation is constrained.
- General notation—the H matrix and R vector in the relation $Hu = R$ is specified.

$$H\begin{bmatrix} w \\ \theta_t \\ \theta_n \end{bmatrix} = R$$

Note: For the simply supported, fixed, and rotational constrained condition the normal rotation is set to zero, resulting in better convergence.

For other coordinate systems the **Constraint** page looks like any other **Constraint** page in the Structural Mechanics Module.



The constraint can be described using standard or general notation. You select the type of notation using the **Standard notation** button and the **General notation, Hu=R** button.

In standard notation you constrain the displacement and rotations independently. The check box in front of $\mathbf{R_z}$, $\mathbf{R_x}$, and $\mathbf{R_y}$ activates the constraint, and you can then enter the value or expression for the displacement in the edit fields. The default value is `0`.

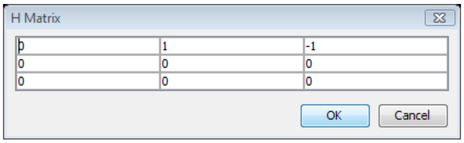
In general notation, the H matrix and R vector in the relation

$$H\begin{bmatrix} w \\ \theta_x \\ \theta_y \end{bmatrix} = R$$

make it possible to specify constraints as any linear combination of displacement and rotation component. The H matrix and R vector are entered in special matrix dialog boxes by clicking the corresponding **Edit** buttons. For example, you can achieve the condition $\theta_x = \theta_y$ using the settings

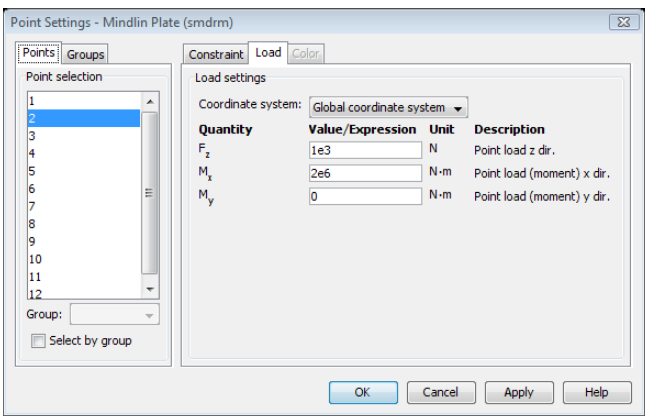
$$
H = \begin{bmatrix} 0 & 1 & -1 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix}, \qquad R = \begin{bmatrix} 0 \\ 0 \end{bmatrix}
$$

The **H Matrix** dialog box for the above example is



Loads

Load is a general name for forces and moments applied to the structure. You can specify loads on all domain types. To do so, click the **Load** tab in the **Subdomain Settings**, **Boundary Settings**, and **Point Settings** dialog boxes. The following picture shows the **Point Settings** dialog box, but the page looks similar on all domain levels.



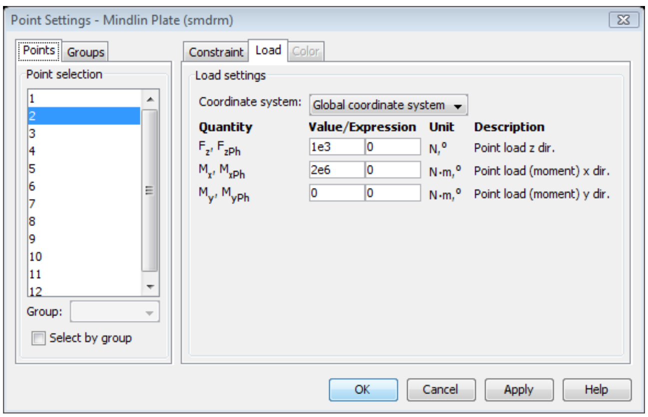
In the subdomain and boundary domains dialog boxes you have an option to specify the load in different ways using the thickness. The loads can be defined on different domains in the following way. The SI unit is shown in parenthesis.

POINT	BOUNDARY	SUBDOMAIN
force (N), moment (Nm)	force/area (N/m^2), moment/area (N/m) or force/length (N/m), moment/length (N)	force/volume (N/m^3), moment/volume (N/m^2) or force/area (N/m^2), moment/area (N/m)

With the **Coordinate system** list you control in what coordinate system the load is defined. Available options are:

- Global coordinate system
- Tangential and normal coordinate system, only available on boundaries
- User-defined coordinate systems, if there are any local coordinate systems defined. Read more about creation of coordinate system in the coordinate system section.

The frequency response analysis type requires additional input. The analysis type is controlled from the **Application Mode Properties** dialog box. When frequency response is selected as analysis type, the **Load** page changes appearance:



For frequency response analysis the harmonic load is split into 3 different parameters:

- The amplitude value, F
- The amplitude factor, F_{Amp} (a dimensionless number; the default value is 1)
- The phase (F_{Ph}).

Together they define a harmonic load whose amplitude and phase shift can vary with the excitation frequency, f:

$$F_{\mathrm{freq}} = F \cdot F_{\mathrm{Amp}}(f) \cdot \cos(2\pi f + F_{\mathrm{Ph}}(f))$$

On subdomains additional options are available controlling if and how thermal strains should be included in the analysis. They are explained in the next section.

Thermal Coupling

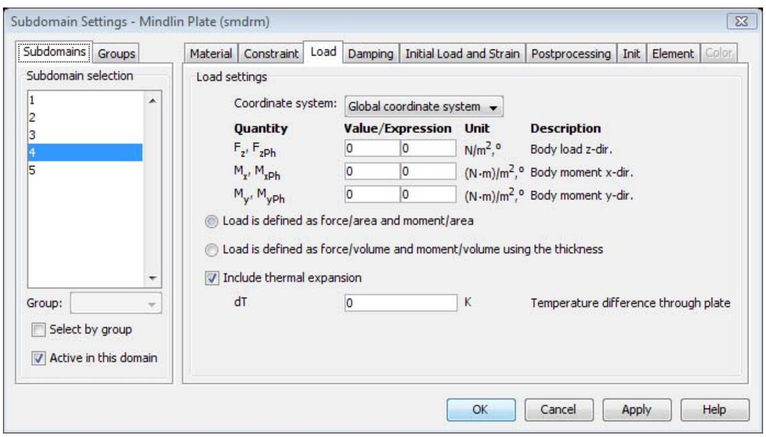
Material expands with temperature, which causes thermal strains to develop in the material. The plate can only handle temperature difference through the plate. The thermal strains together with the initial strains and elastic strains from structural loads form the total strain.

$$\varepsilon = \varepsilon_{\mathrm{el}} + \varepsilon_{\mathrm{th}} + \varepsilon_i$$

where

$$\varepsilon_{\mathrm{th}} = \alpha \Delta T \frac{z}{\mathrm{th}}$$

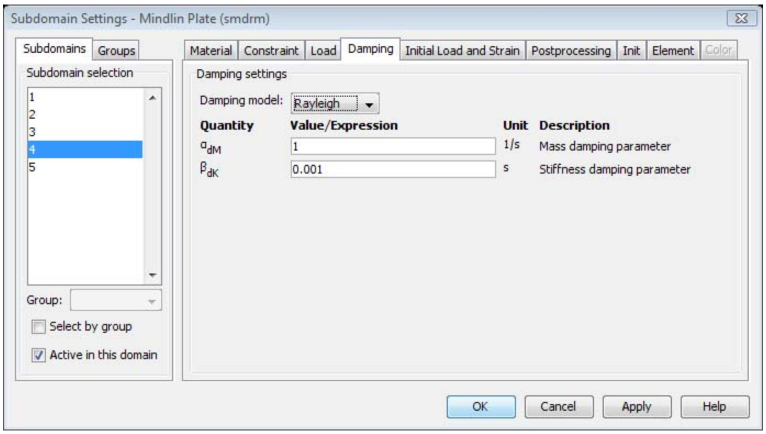
Thermal coupling means that the thermal expansion is included in the analysis. For details on thermal coupling, see page 234. Thermal effects are specified on the **Load** page in the **Subdomain Settings** dialog box.



The **Include thermal expansion** check box adds thermal effects. In the **dT** edit field the temperature difference through the plate, ΔT is specified. The thermal expansion coefficient is specified on the **Material** page described in "Material" on page 240. ΔT can be any expression and is typically another variable solved for in a COMSOL Multiphysics heat transfer application mode. The temperature coupling can be used in any type of analysis.

Damping

In transient and frequency response analyses you have the possibility to model undamped or damped problems. In the Structural Mechanics Module you can specify damping on the subdomain level using the **Damping** page that appears in the **Subdomain Settings** dialog box. From the **Damping models** list you can select **No damping**, **Rayleigh**, or **Loss factor**, and the layout of the dialog box changes for each model.



Damping page when Rayleigh damping is selected.

Note: Loss factor damping is valid only for frequency response analysis. If you choose transient analysis and loss factor damping, COMSOL Multiphysics solves the model with no damping.

Table 8-1 and the subsequent text describe the parameters that define damping:

TABLE 8-1: PARAMETERS FOR DAMPING MODELS

PARAMETER	VARIABLE	DESCRIPTION	DAMPING MODEL
α_{dM}	`alphadM`	Mass-damping parameter	Rayleigh
β_{dK}	`betadK`	Stiffness-damping parameter	Rayleigh
η	`eta`	Loss factor	Loss factor

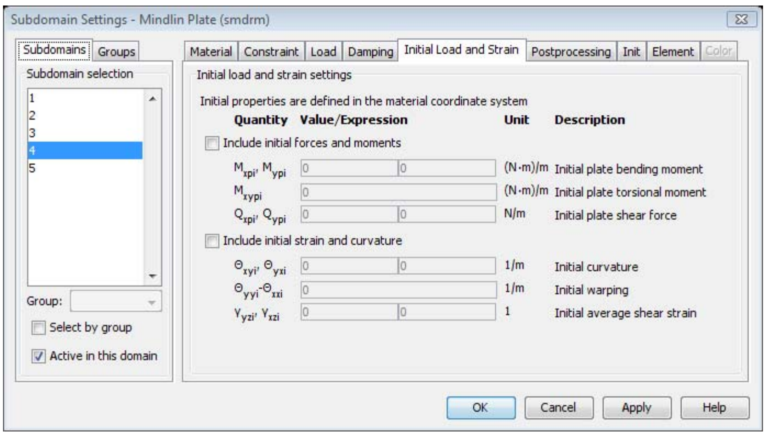
Mass damping parameter It defines the Rayleigh damping model's mass damping, α_{dM}.

Stiffness damping parameter It defines the Rayleigh damping model's stiffness damping, β_{dK}.

Loss factor It defines the loss factor η for the loss factor damping model.

Initial Load and Strain

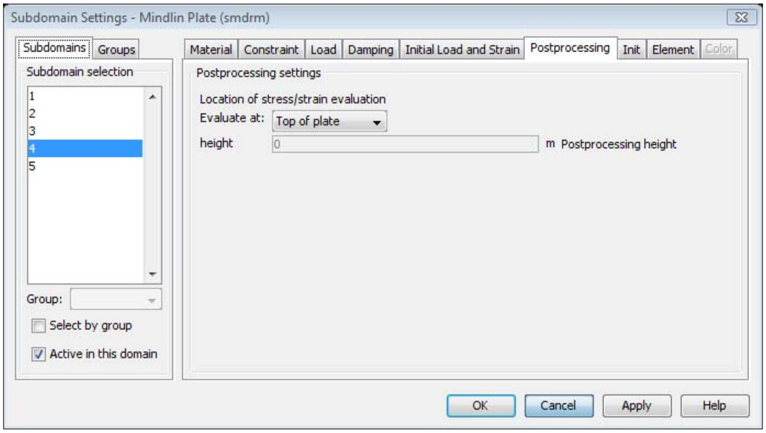
Initial stress and strain can be included in the analysis. For the plate formulation this transforms to initial internal plate moments and shear forces and initial curvatures and initial average shear strains. Initial load and strain can be viewed as different ways to express the same thing. Initial load and strain are specified on the **Initial Load and Strain** page in the **Subdomain Settings** dialog box.



The option to include initial forces and moments and initial curvature and strain is controlled independently using the two check boxes **Include initial forces and moments** and **Include initial strain and curvature**.

Postprocessing

The predefined postprocessing variables include: all non-zero stress and strain tensor components, principal stresses and strains, in-plane and out-of-plane forces, bending and torsional moments, and von Mises and Tresca effective stresses. The stress and strain tensor components and effective stresses can be evaluated at an arbitrary distance from the mid surface. This height is controlled from the **Postprocessing** page in the **Subdomain Settings** dialog box.



With the **Evaluate at** list you control where the stress and strain should be evaluated, available options are:

- Top of plate (default)
- Midplane of plate
- Bottom of plate
- Specified height

Select **Specified height** to specify a postprocessing height explicitly using the **height** edit field.

The displacement and rotations in radians and, for a transient analysis, the velocity and angular velocity can be plotted.

9

Beams

A beam is a slender structure which is assumed to be fully described by the properties—area, moments of inertia, density—of the cross section. Beams are the choice for modeling reinforcements in 3D solids and shell structures, as well as in 2D solids under the plane stress assumption. Naturally, they can also model lattice works, both planar and three-dimensional.

Beams can sustain loads and moments in any direction, both distributed and on individual nodes. The beam's ends and interconnections can be free, simply supported, or clamped. In fact, the simplified boundary conditions are usually responsible for most of the difference that may be found between a beam solution and a full 3D solid simulation of the same structure. Point constraints are well-behaved, in contrast to the solid case. Discrete point masses and mass moments of inertia can be used.

The Structural Mechanics Module's beam application modes are based on the principle of virtual work. The resulting equation can equivalently be viewed as a weak formulation of an underlying PDE. The beam application modes use special shape function classes to define stresses and strains, which are used in setting up the weak form equation.

Theory Background

Shape functions

The degrees of freedom are defined by a shape function object, different shape functions for the in-plane and 3D Euler beam.

IN-PLANE EULER BEAM

The shape function object for the in-plane Euler beam application mode is

```
appl.shape{1}=sheulbps('u','v','th')
```

The shape function class defines the following variables:

VARIABLE	NAME	DESCRIPTION
u	u	Global displacement in x direction
v	v	Global displacement in y direction
θ	th	Rotation angle about an axis pointing out from the xy-plane (global z-axis)
$\frac{\partial\theta}{\partial s}$	ths	Tangential derivative along the edge direction of the rotation angle θ
$\frac{\partial^2\theta}{\partial s^2}$	thss	The second tangential derivative of the rotation angle θ
$\frac{\partial u_{\mathrm{axi}}}{\partial s}$	uvts	The tangential derivative of the axial displacement in the edge direction

See `sheulbps` on page 166 of the *Structural Mechanics Module Reference Guide* for details.

3D EULER BEAM

The shape function object for the 3D Euler beam application mode is:

```
appl.shape{1}=sheulb3d('u','v','w','thx','thy','thz',point)
```

The shape function class defines the following variables.

VARIABLE	NAME	DESCRIPTION
u	u	Global displacement in x direction
v	v	Global displacement in y direction

VARIABLE	NAME	DESCRIPTION
w	`w`	Global displacement in z direction
θ_x	`thx`	Rotation angle about the global x-axis
θ_y	`thy`	Rotation angle about the global y-axis
θ_z	`thz`	Rotation angle about the global z-axis
u_l	`ul`	Displacement in local x direction
v_l	`vl`	Displacement in local y direction
w_l	`wl`	Displacement in local z direction
θ_{xl}	`thxl`	Rotation angle about the local x-axis
θ_{yl}	`thyl`	Rotation angle about the local y-axis
θ_{zl}	`thzl`	Rotation angle about the local z-axis
$\frac{\partial\theta_{xl}}{\partial s}$	`thxs`	Tangential derivative along the edge direction of the rotation angle about the local x-axis
$\frac{\partial\theta_{yl}}{\partial s}$	`thys`	Tangential derivative along the edge direction of the rotation angle about the local y-axis
$\frac{\partial^2\theta_{yl}}{\partial s^2}$	`thyss`	The second tangential derivative of the rotation angle about the local y-axis
$\frac{\partial\theta_{zl}}{\partial s}$	`thzs`	Tangential derivative along the edge direction of the rotation angle about the local z-axis
$\frac{\partial^2\theta_{zl}}{\partial s^2}$	`thzss`	The second tangential derivative of the rotation angle about the local z-axis
$\frac{\partial u_{\text{axi}}}{\partial s}$	`uvwts`	The tangential derivative of the axial displacement in the edge direction

See `sheulb3d` on page 162 of the *Structural Mechanics Module Reference Guide* for details.

Strain-Displacement/Rotation Relation

The axial strain depends on the rotation derivative and axial displacement derivative defined by the shape function and the z coordinate in the beam. For the 2D case it becomes

$$\varepsilon = z\frac{\partial\theta}{\partial s} + \frac{\partial u_{\mathrm{axi}}}{\partial s}$$

The total strain ε consists of thermal ($\varepsilon_{\mathrm{th}}$), initial ($\varepsilon_{\mathrm{i}}$), and elastic strains($\varepsilon_{\mathrm{el}}$)

$$\varepsilon = \varepsilon_{\mathrm{el}} + \varepsilon_{\mathrm{th}} + \varepsilon_{\mathrm{i}}$$

For the 3D case there are two rotational derivatives.

Stress-Strain Relation

The stress-strain relation in the beam is described by

$$\sigma = E\varepsilon$$

The stress-strain relation for linear conditions including initial stress and strain and thermal effects reads:

$$\sigma = E\varepsilon_{\mathrm{el}} + \sigma_{\mathrm{i}} = E(\varepsilon - \varepsilon_{\mathrm{th}} - \varepsilon_{\mathrm{i}}) + \sigma_{\mathrm{i}}$$

where $\boldsymbol{E}$ is known as Young's modulus or the modulus of elasticity.

Thermal Strain

The temperature is assumed to vary linear across the beam's cross section. For the in-plane beam it becomes

$$T = T_m + \Delta T\frac{z}{h_z}$$

The thermal strain as a function of the z-coordinate and the temperature gradient is

$$\varepsilon_{\mathrm{th}} = \alpha\left(T_m + \Delta T\frac{z}{h_z} - T_{\mathrm{ref}}\right)$$

In the 3D beam the temperature depends on both y and z:

$$T = T_m + \Delta T_z\frac{z}{h_z} + \Delta T_y\frac{y}{h_y}$$

$$\varepsilon_{\mathrm{th}} = \alpha\left(T_m + \Delta T_z\frac{z}{h_z} + \Delta T_y\frac{y}{h_y} - T_{\mathrm{ref}}\right)$$

Initial Load and Strain

The initial stress means the stress before any loads, displacements, and initial strains have been applied.

The initial stress distribution is given as initial moment and initial normal force, for the 2D beam

$$M_{\mathrm{i}} = \int_A \sigma_{\mathrm{i}} z \, dA \qquad N_{\mathrm{i}} = \int_A \sigma_{\mathrm{i}} \, dA$$

In 3D there is an additional bending moments and a torsional moment.

The initial strain is the strain before any loads, displacements, and initial stresses have been applied. The initial strain distribution is given as initial curvature and initial axial strain, for the 2D beam

$$\varepsilon_{\mathrm{i}} = z\left(\frac{\partial\theta}{\partial s}\right)_{\mathrm{i}} + \left(\frac{\partial u_{\mathrm{axi}}}{\partial s}\right)_{\mathrm{i}}$$

In 3D there are two initial rotational derivatives and an initial torsional derivative.

Implementation

The implementation is based on the principle of virtual work, which states that the sum of virtual work from internal strains and external loads equals zero:

$$\delta W = \delta \int_V (-\varepsilon_{\mathrm{el}} \sigma + \mathbf{u}^T \mathbf{F} dV) = 0$$

In 2D the beam moment is defined as

$$M = \int_A \sigma z \, dA = \int_A z(E\varepsilon_{\mathrm{el}} + \sigma_{\mathrm{i}}) dz =$$

$$\int_A z\left(E\left[\left(z\frac{\partial\theta}{\partial s} + \frac{\partial u_{\mathrm{axi}}}{\partial s}\right) - \left(z\left(\frac{\partial\theta}{\partial s}\right)_{\mathrm{i}} + \left(\frac{\partial u_{\mathrm{axi}}}{\partial s}\right)_{\mathrm{i}}\right) - \right.\right.$$

$$\left.\left.\alpha\left(T_m + \Delta T\frac{z}{h_z} - T_{\mathrm{ref}}\right)\right] + \sigma_{\mathrm{i}}\right) dz =$$

$$\int_A z^2\left(E\left[\frac{\partial\theta}{\partial s} - \left(\frac{\partial\theta}{\partial s}\right)_{\mathrm{i}} - \alpha\frac{\Delta T}{h_z}\right]\right) dA + \int_A \sigma_{\mathrm{i}} z \, dz = EI_{yy}\left[\frac{\partial\theta}{\partial s} - \left(\frac{\partial\theta}{\partial s}\right)_{\mathrm{i}} - \alpha\frac{\Delta T}{h_z}\right] + M_{\mathrm{i}}$$

In 3D there is an additional bending moment and torsional moment.

The torsion of the beam is defined using a torsional constant J given by

$$J = \frac{M}{G\theta}l$$

In a similar way as for the bending part a torsional moment is defined as

$$M_{x\mathrm{l}} = GJ\left(\frac{\partial\theta_{x\mathrm{l}}}{\partial s} - \left(\frac{\partial\theta_{x\mathrm{l}}}{\partial s}\right)_{\mathrm{i}}\right) + M_{x\mathrm{i}}$$

The normal force is defined as

$$\begin{aligned}
&N = \int_A \sigma dA = \int_A (E\varepsilon_{\mathrm{el}} + \sigma_{\mathrm{i}})dz = \\
&\int_A \left(E\left[\left(z\frac{\partial\theta}{\partial s} + \frac{\partial u_{\mathrm{axi}}}{\partial s}\right) - \left(z\left(\frac{\partial\theta}{\partial s}\right)_{\mathrm{i}} + \left(\frac{\partial u_{\mathrm{axi}}}{\partial s}\right)_{\mathrm{i}}\right) - \alpha\left(T_m + \Delta T\frac{z}{h_z} - T_{\mathrm{ref}}\right)\right] + \sigma_{\mathrm{i}}\right)dz = \\
&\int_A \left(E\left[\left(\frac{\partial u_{\mathrm{axi}}}{\partial s} - \left(\frac{\partial u_{\mathrm{axi}}}{\partial s}\right)_{\mathrm{i}}\right) - \alpha(T_m - T_{\mathrm{ref}})\right]\right)dA + \int_A \sigma_{\mathrm{i}} dz = \\
&EA\left[\left(\frac{\partial u_{\mathrm{axi}}}{\partial s} - \left(\frac{\partial u_{\mathrm{axi}}}{\partial s}\right)_{\mathrm{i}}\right) - \alpha(T_m - T_{\mathrm{ref}})\right] + N_{\mathrm{i}}
\end{aligned}$$

Using the beam moment and normal force the expression for the virtual work becomes very compact, for the 2D beam it becomes

$$\delta W = \int_L \left(M\left(\frac{\partial\theta}{\partial s}\right)_{\mathrm{test}} + N\left(\frac{\partial u_{\mathrm{axi}}}{\partial s}\right)_{\mathrm{test}}\right)dx$$

For 3D it becomes

$$\delta W = \int_L \left(M_{y\mathrm{l}}\left(\frac{\partial\theta_{y\mathrm{l}}}{\partial s}\right)_{\mathrm{test}} + M_{z\mathrm{l}}\left(\frac{\partial\theta_{z\mathrm{l}}}{\partial s}\right)_{\mathrm{test}} + N\left(\frac{\partial u_{\mathrm{axi}}}{\partial s}\right)_{\mathrm{test}} + M_{x\mathrm{l}}\left(\frac{\partial\theta_{x\mathrm{l}}}{\partial s}\right)_{\mathrm{test}}\right)dx$$

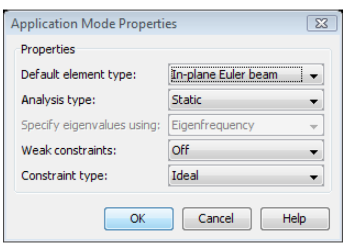
Application Mode Description

This section describe how to define a beam model. It is divided into the following sections:

- Properties
- Scalar Variables
- Material
- Cross Section
- Constraint
- Load
- Discrete Mass
- Thermal Coupling
- Initial Load and Strain
- Postprocessing

Properties

To open **Application Mode Properties** dialog box, choose **Physics>Propertries**.



In the **Application Mode Properties** dialog box you control global settings for the model.

- **Analysis type**: A list of different analyses to perform. It affects both the equations and what solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. The available analysis types use the following solvers.

ANALYSIS TYPE	COMSOL MULTIPHYSICS SOLVER
Static	Stationary
Eigenfrequency	Eigenvalue

ANALYSIS TYPE	COMSOL MULTIPHYSICS SOLVER
Time dependent	Time dependent
Frequency response	Parametric
Parametric	Parametric
Quasi-static transient	Time dependent

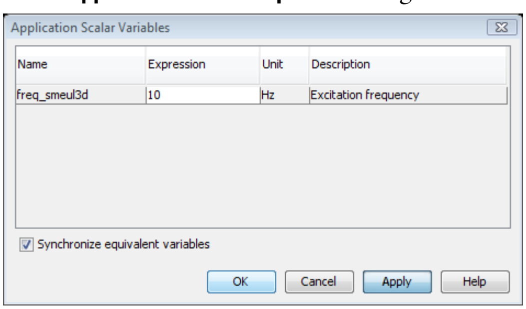
- **Weak constraints**: Controls whether or not weak constraints are active. Use weak constraints for accurate reaction-force computation. When weak constraints are enabled, all constraints are weak by default, but it is possible to change this setting for individual domains.
- **Constraint type**: Constraints can be ideal or nonideal (see "Ideal vs. Non-Ideal Constraints" on page 301 in the *COMSOL Multiphysics Modeling Guide*).

Scalar Variables

There are two different scalar variables:

- Excitation frequency, `freq`, which is applicable only for frequency response analysis.
- Complex angular frequency, `jomega`, which is applicable only for eigenfrequency analysis. You normally do not need to edit the complex angular frequency.

The **Scalar Variables** menu item on the **Physics** menu is enabled only when you have selected **Frequency Response, Damped Eigenfrequency**, or **Eigenfrequency** as **Analysis type** in the **Application Mode Properties** dialog box.



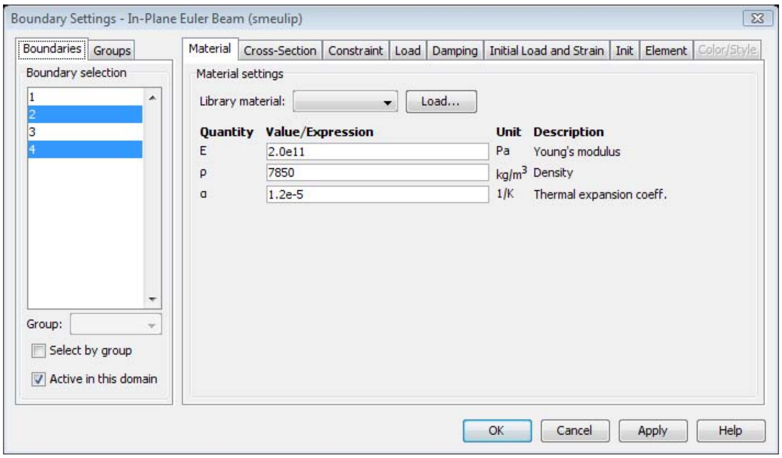
The excitation frequency is the frequency of the harmonic loads in a frequency response analysis.

When you have selected frequency response as the analysis type, the default solver is the parametric solver making it easy to perform a frequency sweep over several excitation frequencies in a single analysis. In this case, enter `freq_smeul3d` in the

Parameter name edit field on the **Parametric** page in the **Solver Parameters** dialog box. The values entered in the **Parameter values** edit field then override the excitation frequency entered in the **Application Scalar Variables** dialog box.

Material Properties

The material properties are defined on the **Material** page in the **Boundary Settings** dialog box for the 2D in-plane Euler beam and in the **Edge Settings** dialog box for the 3D Euler beam.



The material properties are shown in the table below.

PARAMETER	VARIABLE	DESCRIPTION	COMMENT
E	E	Young's modulus	
ν	nu	Poisson's ratio	Only 3D Euler beam
ρ	rho	Density	
α	alpha	Thermal expansion coefficient	

Young's modulus Defines the modulus of elasticity, E of the material. It is the spring stiffness in Hooke's law, shown below in 1D form

$$\sigma = E\varepsilon$$

where σ is the stress and ε is the strain.

Poisson's ratio Denoted by ν, defines the normal strain in the perpendicular direction, generated from a normal strain in the other direction.

$$\varepsilon_{\perp} = -\upsilon\varepsilon_{||}$$

Used to calculate the shear modulus G, used in the torsional part of the 3D Euler beam.

Density This material property, ρ, specifies the density of the material.

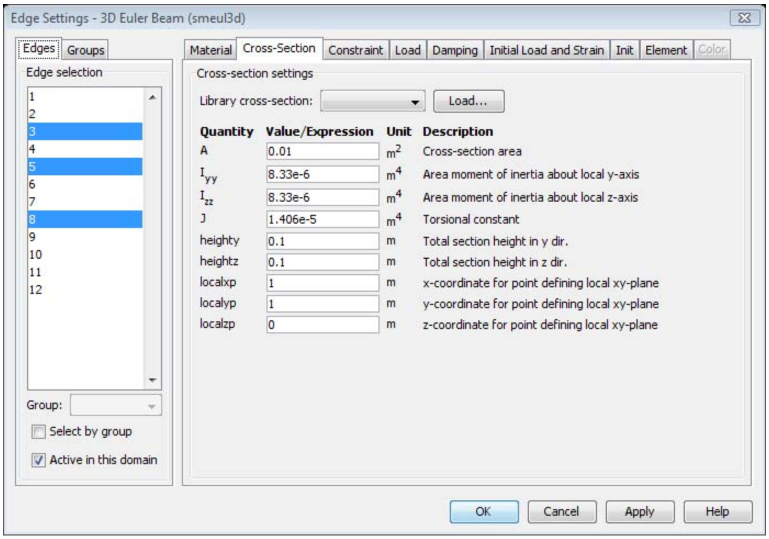
Thermal expansion coefficient Defines how much a material expands due to an increase in temperature.

$$\varepsilon_{\mathrm{th}} = \alpha\left(T_m + \Delta T_z\frac{z}{h} + \Delta T_y\frac{y}{h} - T_{\mathrm{ref}}\right)$$

where $\varepsilon_{\mathrm{th}}$ is the thermal strain, ΔT_y and ΔT_z are the temperature difference over the cross section of the beam in the y and z directions, and α is the thermal expansion coefficient. T_m is the temperature in the middle and T_{ref} is the stress free reference temperature.

Cross Section

The cross-sectional properties are defined on the **Cross-Section** page in the **Edge Settings** or **Boundary Settings** dialog box.



The following table lists the cross-section properties:

PARAMETER	VARIABLE	DESCRIPTION	COMMENT
A	`A`	Cross-sectional area	
I_{yy}	`Iyy`	Area moment of inertia about local y-axis	
I_{zz}	`Izz`	Area moment of inertia about local z-axis	3D Euler beam only
J	`J`	Torsional constant	3D Euler beam only
height$_y$ (h_y)	`heighty`	Total section height in the y direction	
height$_z$ (h_z)	`heightz`	Total section height in the z direction	3D Euler beam only
localxp	`localxp`	x-coordinate for point defining local xy-plane	3D Euler beam only
localyp	`localyp`	y-coordinate for point defining local xy-plane	3D Euler beam only
localzp	`localzp`	z-coordinate for point defining local xy-plane	3D Euler beam only

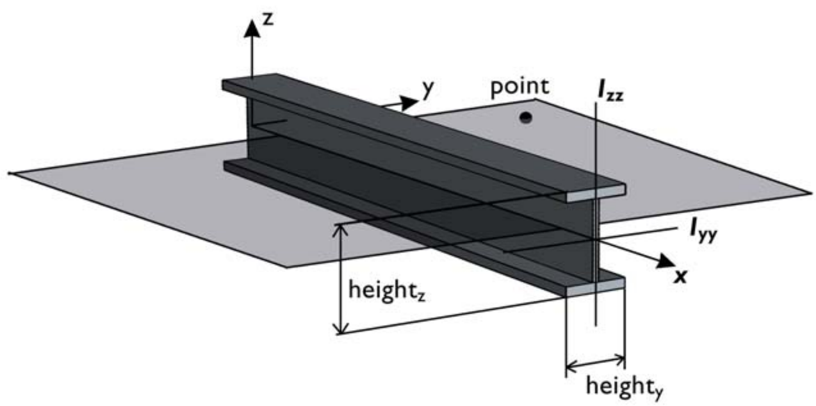
3D beams modeled on edges need a local coordinate system for a number of reasons:

- Input data—you need a coordinate system to specify input data such as area moment of inertia.

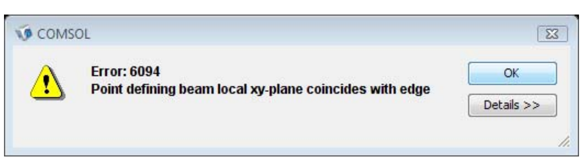
- Postprocessing—if you want to look at bending moments or shear forces, you must know the direction of the coordinate system.
- Loads—if you want to apply loads in a local coordinate system you need to be able to specify it.

If your beam's cross section is a square or circle (solid or tube), the area moments of inertia are the same independent of direction, so the beam is totally symmetric and you do not need to worry about the local coordinate system unless you are interested in looking at results defined using the local coordinate system. Such results are bending moments, shear forces, local displacements and rotations.

The coordinate system is defined in the following way. The *x* direction is in the edge direction. The positive edge direction can be checked by plotting the edge arc length parameter `s1` and see in what direction it increases. You can also plot the tangential variable `t1x`, `t1y`, and `t1z` to check the direction of the edge. The coordinates of an additional point (`localxp`, `localyp`, and `localzp`), specified on the **Cross-Section** page in the **Edge Settings** dialog box, defines the local *xy*-plane with the positive *y* direction defined so that the point lies in the positive quadrant. See the previous dialog box and the following figure.



For the creation of a local coordinate system to be possible, the point cannot coincide with the edge or the edge extension. If you do this you get an error message: **Point defining beam local xy-plane coincides with edge**.



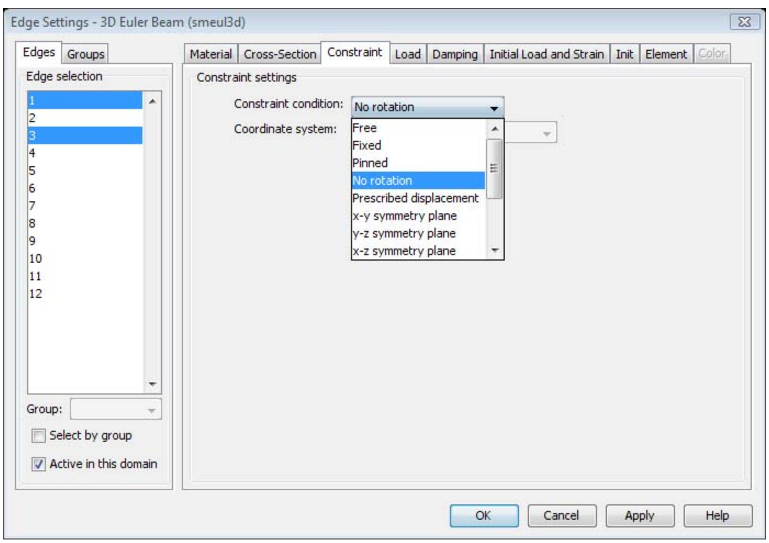
This error might occur even if you have a totally symmetric cross section and do not have to worry about the direction. If this happens you must enter coordinates far away from your edge and not along the edge extension. You can do this for a number of edges at the same time using a point far away from the geometry.

Note: The default settings for the global coordinates of the point are `[1,1,0]`.

Usually a number of edges in a plane have the same orientation. It is then easy to select all edges and specify a point anywhere in the same plane, not coinciding with an edge or an edge extension.

Constraint

A constraint specifies the displacement and rotation of a certain part of the beam. Constraints can be defined on all valid domain levels such as points, edges, and boundaries. The constraint is controlled from the **Constraint** page in the **Boundary Settings**, **Edge Settings**, and **Point Settings** dialog boxes.



An example of a beam application mode Constraint page, taken here from the 3D Euler Beam application mode Points Settings dialog box.

The figure shows the **Constraint** page of the **Edge Settings** dialog box for the 3D Euler beam application mode. The page looks similar on all domain levels in both of the beam application modes, differing only regarding the variables to constrain:

- For the 2D Euler beam, two displacement and one rotation.
- For the 3D Euler beam, three displacements and three rotations.

Within the dialog box, the **Constraint condition** list lets you control what type of constraint you want to define. You have the following options to choose between:

CONSTRAINT CONDITION	POINT	BOUNDARY/ EDGE	USE WHEN
Free	√	√	The domain has no constraint
Pinned	√	√	The displacement in the domain is fixed in all direction
Fixed	√	√	The displacement and rotations in the domain are fixed in all directions
No rotation	√	√	The rotations in the domain are fixed in all directions
Prescribed displacement	√	√	The displacement or rotation in any direction need to be prescribed
Symmetry plane		√ (2D only)	The boundary is a symmetry plane
x-y symmetry plane	√	√	The selected coordinate system's xy-plane is a symmetry plane
y-z symmetry plane	√	√	The selected coordinate system's yz-plane is a symmetry plane
x-z symmetry plane	√	√	The selected coordinate system's xz-plane is a symmetry plane
Antisymmetry plane		√ (2D only)	The boundary is an antisymmetry plane
x-y antisymmetry plane	√	√	The selected coordinate system's xy-plane is an antisymmetry plane
y-z antisymmetry plane	√	√	The selected coordinate system's yz-plane is an antisymmetry plane
x-z antisymmetry plane	√	√	The selected coordinate system's xz-plane is an antisymmetry plane
Prescribed velocity	√	√	The velocity and angular velocity in any direction need to be prescribed, only available for frequency response analysis
Prescribed acceleration	√	√	The acceleration or angular acceleration in any direction need to be prescribed, only available for frequency response analysis

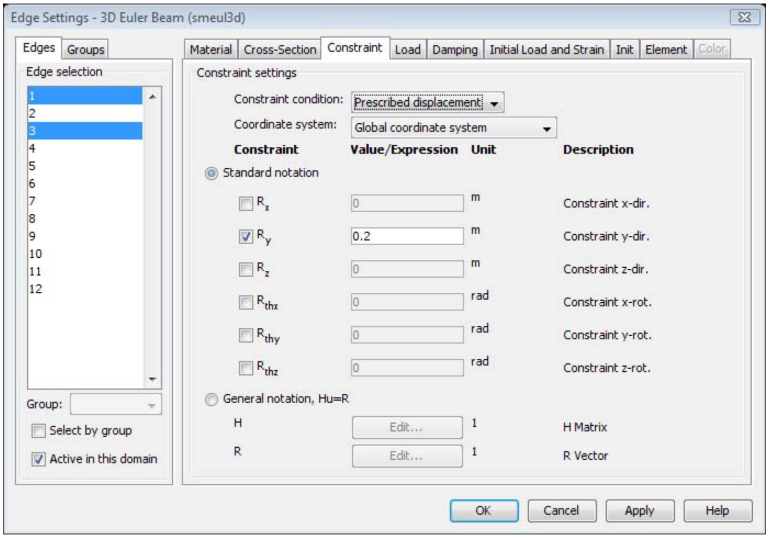
The symmetry or antisymmetry condition has the following interpretation.

CONDITION	X-DISP.	Y-DISP.	Z-DISP.	X-ROT.	Y-ROT.	Z-ROT.
x-y symmetry plane			√	√	√	
y-z symmetry plane	√				√	√
x-z symmetry plane		√		√		√
x-y antisymmetry plane	√	√				√
y-z antisymmetry plane		√	√	√		
x-z antisymmetry plane	√		√		√	

With the **Coordinate system** list you control in what coordinate system the constraint is defined. Available options are:

- Global coordinate system
- Tangential and normal coordinate system, only available on boundaries for the 2D Euler beam.
- User-defined coordinate systems, if there are any local coordinate systems defined. Read more about creation of coordinate system in the coordinate system chapter.
- Beam local coordinate system, only available on edges for the 3D Euler beam.

When you select **Prescribed displacement** a number of new option appears in the dialog box and the **Constraint** page takes on this appearance:



The Constraint page showing the Prescribed displacement options.

You can prescribe a constraint in two ways:

- In standard notation you constrain each displacement direction independently. The check box in front of $\mathbf{R_x}$, $\mathbf{R_y}$, $\mathbf{R_z}$, $\mathbf{R_{thx}}$, $\mathbf{R_{thy}}$, and $\mathbf{R_{thz}}$ activates the constraint, the value/expression of the displacement/rotation can then be entered in the edit fields. The default value is `0`.
- In general notation, the H matrix and R vector in the relation

$$Hu = R$$

make it possible to specify constraints as any linear combination of the available variables.

For the 2D Euler beam application mode the relation is

$$H\begin{bmatrix} u \\ v \\ \theta \end{bmatrix} = R$$

For the 3D Euler beam application mode the relation is

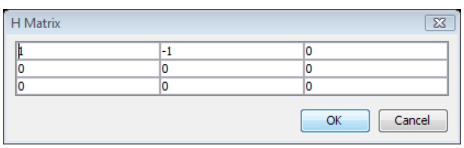
$$
H \begin{bmatrix} u \\ v \\ w \\ \theta_x \\ \theta_y \\ \theta_z \end{bmatrix} = R
$$

The H matrix and R vector are entered in special matrix dialog boxes by clicking the corresponding **Edit** buttons. For example the condition $u = v$ in the 2D Euler beam application mode can be achieved using the settings

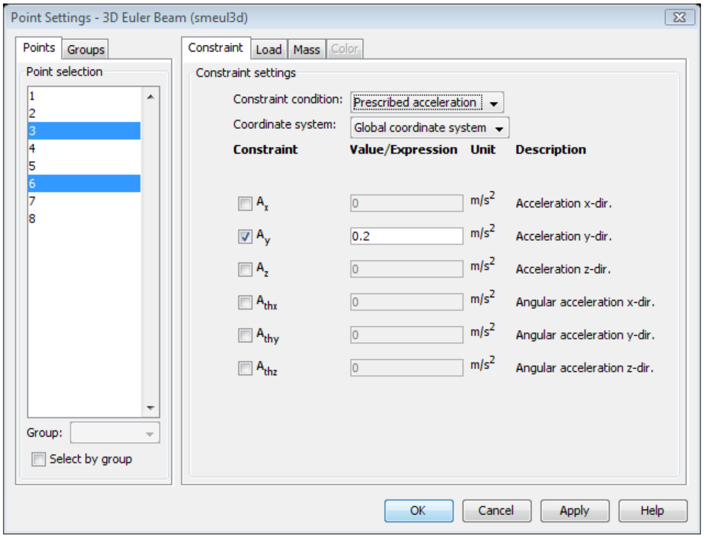
$$
H = \begin{bmatrix} 1 & -1 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} \qquad R = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix}
$$

which force the domain to move only diagonally in the xy-plane.

The **H Matrix** dialog box for the above example is



In a frequency response analysis you have the possibility to specify not only a harmonic displacement and rotation but also a harmonic velocity/angular velocity or acceleration/angular acceleration. You specify the **Prescribed velocity** and **Prescribed acceleration** in the same way as **Prescribed displacement** using **Standard notation**.

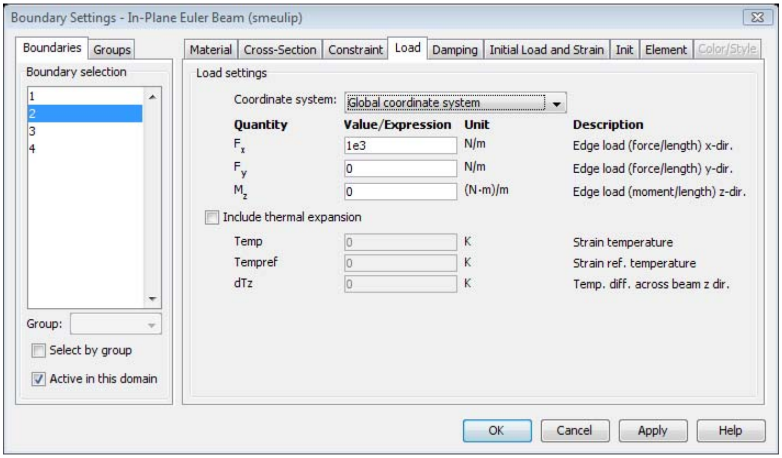


Constraint page showing the Prescribed acceleration settings.

Load

Load is a general name for forces and moments applied to, the structure. Loads can be specified on all domain types. You specify loads on the **Load** page in the **Boundary Settings**, **Edge Settings**, and **Point Settings** dialog boxes. The following picture shows the

Boundary Settings dialog box for the 2D Euler beam application mode, but the page looks similar on all domain levels.



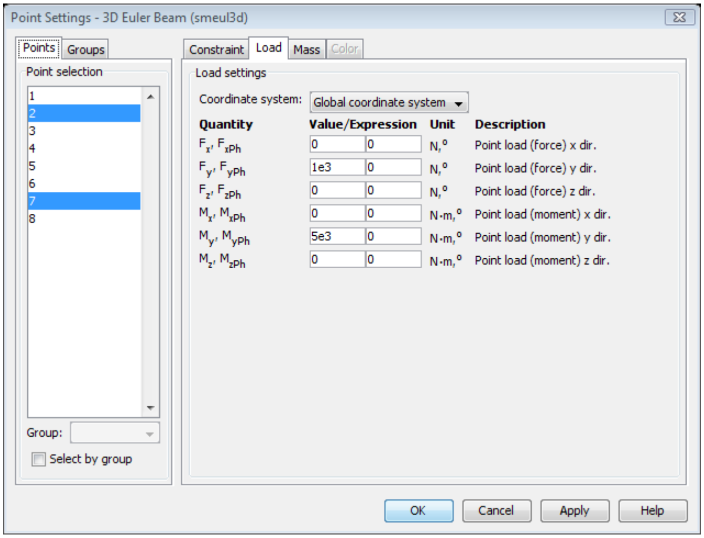
The loads/moments are defined in the following way. The SI unit is shown in parenthesis.

DOMAIN TYPE	LOAD (UNIT)	MOMENT (UNIT)
point	force (N)	moment (Nm)
edge, boundary	force/length (N/m)	moment/length (N)

With the **Coordinate system** list you control in what coordinate system the load is defined. Available options are:

- Global coordinate system
- Tangential and normal coordinate system, only available on boundaries for the 2D Euler beam.
- User-defined coordinate systems, if there are any local coordinate systems defined. Read more about creation of coordinate system in the coordinate system section.
- Beam local coordinate system, only available on edges for the 3D Euler beam.

For the frequency response analysis type, additional input data is specified. You control the analysis type from the **Application Mode Properties** dialog box. If you select frequency response as analysis type, the **Load** page changes appearance to looks like the following image:



For frequency response analysis the harmonic load is split into 3 different parameters:

- The amplitude value (F, M)
- The amplitude value factor (F_{Amp}, M_{Amp}) (a dimensionless number; the default value is 1)
- The phase (F_{Ph}, M_{Ph})

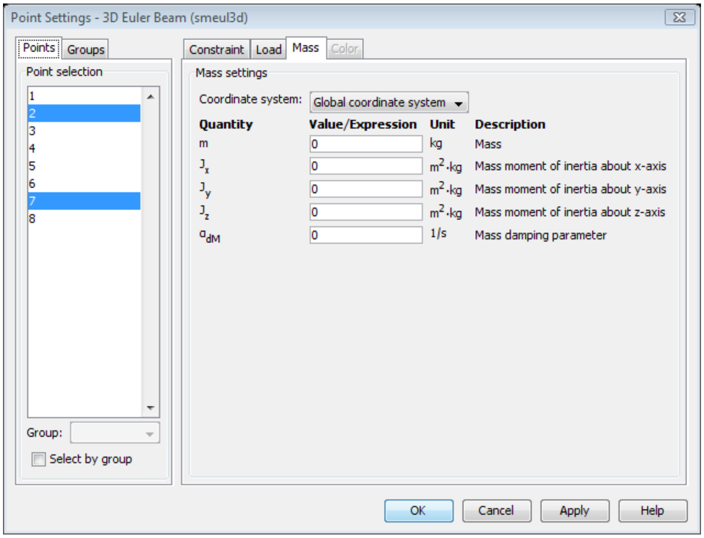
Together they define a harmonic load whose amplitude and phase shift can vary with the excitation frequency f:

$$F_{\mathrm{freq}} = F \cdot F_{\mathrm{Amp}}(f) \cdot \cos(2\pi f + F_{\mathrm{Ph}}(f))$$

On the edge and boundary domain levels additional options are available controlling if and how thermal strains should be included in the analysis. They are explained in the section "Thermal Coupling" on page 274.

Discrete Mass

Discrete mass or mass moment of inertia are concentrated to a point in contrast to distributed mass modeled through the density and area of the beam. You specify discrete mass and mass moment of inertia on the **Mass** page in the **Point Settings** dialog box.



With the **Coordinate system** list you control in what coordinate system the principal mass moment of inertias are defined (only possible for 3D Euler beams). Available options are:

- Global coordinate system.
- User-defined coordinate systems, if there are any local coordinate systems defined. Read more about creation of coordinate system in "Coordinate Systems" on page 144.

The mass properties are shown in the following table:

PARAMETER	VARIABLE	DESCRIPTION	SI UNIT	COMMENT
m	`m`	Mass	kg	
J_x	`Jx`	Mass moment of inertia about x-axis	kg·m^2	Only 3D Euler beam
J_y	`Jy`	Mass moment of inertia about x-axis	kg·m^2	Only 3D Euler beam
J_z	`Jz`	Mass moment of inertia about x-axis	kg·m^2	
α_{dM}	`alphadM`	Mass damping parameter	1/s	

Thermal Coupling

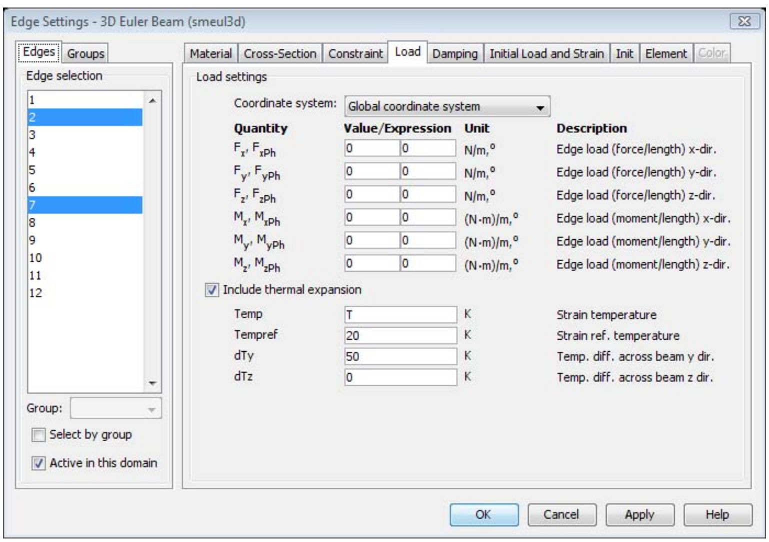
Material expands with temperature, which causes thermal strains to develop in the material. The beams can handle any temperature variation along the beam, and linear variation across the beam. The thermal strains together with the initial strains and elastic strains from structural loads form the total strain.

$$\varepsilon = \varepsilon_{\mathrm{el}} + \varepsilon_{\mathrm{th}} + \varepsilon_{\mathrm{i}}$$

where

$$\varepsilon_{\mathrm{th}} = \alpha\left(T_m + \Delta T_z \frac{z}{h_z} + \Delta T_y \frac{y}{h_y} - T_{\mathrm{ref}}\right)$$

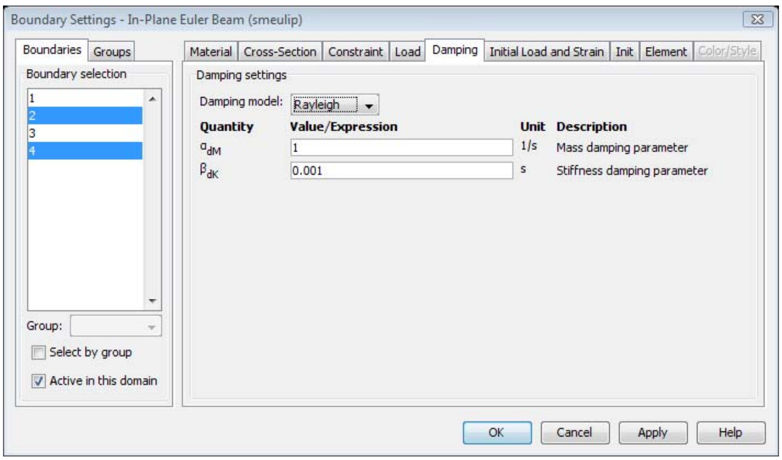
Thermal coupling means that the thermal expansion is included in the analysis. Details on thermal coupling is found on page 256. Thermal effects are specified on the **Load** page in the **Edge Settings** and **Boundary Settings** dialog boxes.



The **Include thermal expansion** check box adds thermal effects. In the **Temp** and **Tempref** edit fields the strain temperature in the middle of the cross section, T_m and stress free reference temperature T_{ref} are specified. In the **dTy** and **dTz** edit fields the temperature difference across the beams cross section, ΔT_y and ΔT_z is specified. The thermal expansion coefficient are specified on the **Material** page, described in the Material section on page 261. T_m, T_{ref}, ΔT_y, and ΔT_z can be any expression and are typically another variable solved for in an application mode. The temperature coupling can be used in any type of analysis.

Damping

In transient and frequency response analyses you have the possibility to model undamped or damped problems. In the Structural Mechanics Module you can specify damping in the subdomain level using the **Damping** page that appears in the **Boundary Settings** (2D) or **Edge Settings** (3D) dialog box. From the **Damping models** list you can select **No damping**, **Rayleigh**, or **Loss factor**, and the layout of the dialog box changes for each model.



Damping page when Rayleigh damping is selected.

Note: Loss factor damping is valid only for frequency response analysis. If you choose transient analysis and loss factor damping, the model will be solved with no damping.

Table 9-1 and the following text describe the parameters that define damping:

TABLE 9-1: PARAMETERS FOR DAMPING MODELS

PARAMETER	VARIABLE	DESCRIPTION	DAMPING MODEL
α_{dM}	`alphadM`	Mass-damping parameter	Rayleigh
β_{dK}	`betadK`	Stiffness-damping parameter	Rayleigh
η	`eta`	Loss factor	Loss factor

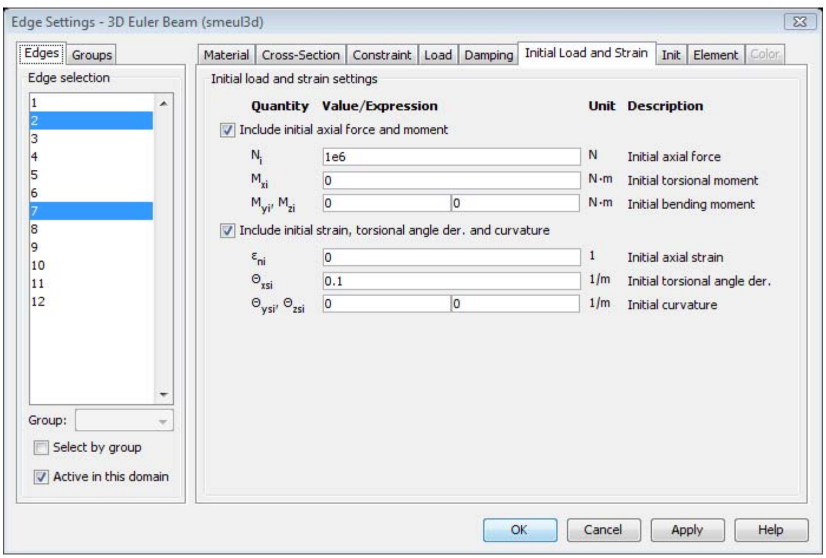
Mass damping parameter Defines the Rayleigh damping model's mass damping, α_{dM}.

Stiffness damping parameter Defines the Rayleigh damping model's stiffness damping, β_{dK}.

Loss factor Defines the loss factor η for the loss factor damping model.

Initial Load and Strain

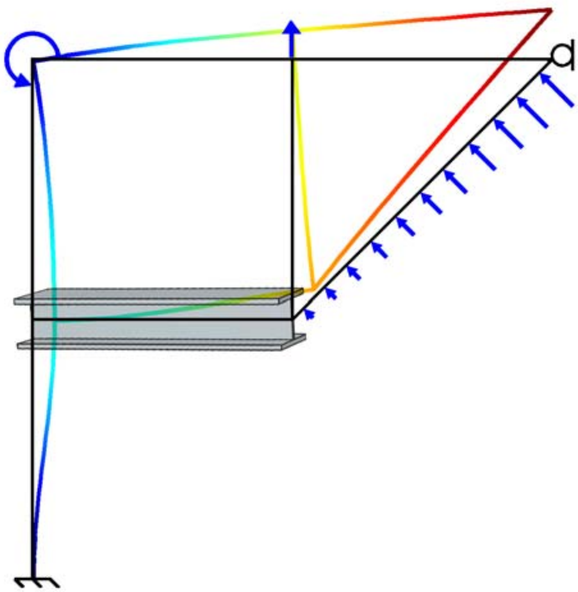
Initial stress and strain can be included in the analysis. For the beam formulations the initial stresses transforms to initial internal beam moments, initial torsional moment, and initial normal force. The initial strains consists of initial curvatures, initial torsional angle derivative, and initial axial strain. Initial load and strain can be viewed as different ways to express the same thing. Initial load and strain are specified on the **Initial Load and Strain** page in the **Edge Settings** and **Boundary Settings** dialog boxes.



The option to include initial forces and moments and initial curvature and strain is controlled independently using the two check boxes **Include initial axial force and moments** and **Include initial strain and curvature**.

In-Plane Euler Beam

Use the In-Plane Euler Beam application mode to analyze planar lattice works of uniaxial beams.



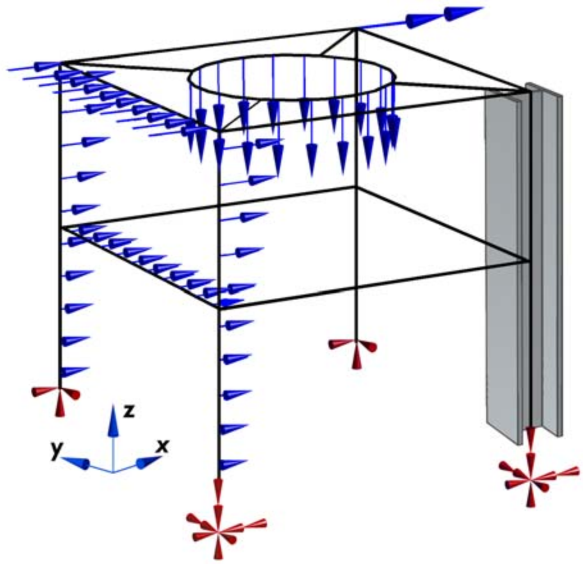
In-plane Euler beams are defined on edges in 2D. All settings for the application mode are described in “Application Mode Description” on page 259.

Variables and Space Dimensions

The degrees of freedom (dependent variables) are the global displacements u and v in the global x and y directions and the rotation θ about the global z-axis.

3D Euler Beam

Use the 3D Euler Beam application mode to model three-dimensional frameworks of uniaxial beams.



3D Euler beams are defined on edges in 3D. All settings for the application mode appear in “Application Mode Description” on page 259.

Variables and Space Dimensions

The degrees of freedom (dependent variables) are the global displacements u, v, w in the global x, y, z directions and the global rotations θ_x, θ_y, and θ_z about the global x-, y-, and z-axes.

10

Trusses

Trusses are elements that can only sustain axial forces. They have displacements as degrees of freedom. Trusses are sometimes referred to as bars or spars. They live on boundaries in 2D and edges in 3D. The truss application modes support the same analysis types as the continuum application modes. You can use trusses to model truss works where the edges are straight but also to model sagging cables like the deformation of a wire exposed to gravity. In such applications trusses are often referred to as cable elements.

Theory Background

Trusses is modeled using Lagrange shape function. The Lagrange shape function makes it possible to specify both normal strains and Green-Lagrange strains to handle small strains as well as large deformations.

Strain-Displacement Relation

The axial strain ε_n is calculated by expressing the global strains in tangential derivatives and projecting the global strains on the edge.

$$\varepsilon_n = \mathbf{t}^t \varepsilon_{gT} \mathbf{t} \tag{10-1}$$

where $\mathbf{t}$ is the edge tangent vector and ε_{gT} is defined as

$$\varepsilon_{gT} = \begin{bmatrix} \varepsilon_{xT} & \varepsilon_{xyT} & \varepsilon_{xzT} \\ \varepsilon_{xyT} & \varepsilon_{yT} & \varepsilon_{yzT} \\ \varepsilon_{xzT} & \varepsilon_{yzT} & \varepsilon_{zT} \end{bmatrix} \tag{10-2}$$

The strains can be expressed as either engineering strains for small displacements or Green strains for large displacements. The Green strain tensor used for large displacements is defined as

$$\varepsilon_{ijT} = \frac{1}{2}\left(\left.\frac{\partial u_i}{\partial x_j}\right|_T + \left.\frac{\partial u_j}{\partial x_i}\right|_T + \left.\frac{\partial u_k}{\partial x_i}\right|_T \cdot \left.\frac{\partial u_k}{\partial x_j}\right|_T\right) \tag{10-3}$$

The engineering strain tensor used for small displacements is defined as

$$\varepsilon_{ijT} = \frac{1}{2}\left(\left.\frac{\partial u_i}{\partial x_j}\right|_T + \left.\frac{\partial u_j}{\partial x_i}\right|_T\right) \tag{10-4}$$

The axial strain written out becomes

$$\begin{aligned} \varepsilon_n = {} & t_x(\varepsilon_{xT}t_x + \varepsilon_{xyT}t_y + \varepsilon_{xzT}t_z) + \\ & t_y(\varepsilon_{xyT}t_x + \varepsilon_{yT}t_y + \varepsilon_{yzT}t_z) + \\ & t_z(\varepsilon_{xzT}t_x + \varepsilon_{yzT}t_y + \varepsilon_{zT}t_z) \end{aligned} \tag{10-5}$$

Stress-Strain Relation

The constitutive relation for the axial stress including thermal strain and initial stress and strain is

$$\sigma_n = E(\varepsilon_n - \alpha(T - T_{\mathrm{ref}}) - \varepsilon_{ni}) + \sigma_{ni} \tag{10-6}$$

Implementation

Using the principle of virtual work results in the following weak formulation

$$\delta W = d\int_V (-\varepsilon_n \sigma_n + \mathbf{u}^t \mathbf{F}_V) dV + \sum_i \mathbf{u}^t \mathbf{F}_{Pi} \tag{10-7}$$

where the summation stands for summation over all points in the geometry. Replacing the integration over the cross section with the cross-sectional area (A) and the volume forces with line forces, the equation becomes

$$\delta W = \int_L (-\varepsilon_{n\mathrm{test}} \sigma_n A + \mathbf{u}^t_{\mathrm{test}} \mathbf{F}_L) dL + \sum_i \mathbf{u}^t_{\mathrm{test}} \mathbf{F}_{Pi} \tag{10-8}$$

Straight Edge Option

The optional constraint to enforce the nodes to lie on the straight line between the end points of the edge are formulated as follows:

Starting with the large displacement case, let $\mathbf{x}_{d1}$ and $\mathbf{x}_{d2}$ be the deformed position of the two end points of the edge

$$\mathbf{x}_{di} = \mathbf{u}_i + \mathbf{x}_i \tag{10-9}$$

where $\mathbf{u}_i$ is the displacement, and $\mathbf{x}_i$ is the coordinate (undeformed position) at end point i. The equation for the straight line through the end points is

$$\mathbf{x} + \mathbf{u} = \mathbf{x}_{d1} + t\mathbf{a} \tag{10-10}$$

where t is a parameter along the line, and $\mathbf{a}$ is the direction vector for the line. $\mathbf{a}$ is calculated from the deformed position of the end points as

$$\mathbf{a} = \mathbf{x}_{d2} - \mathbf{x}_{d1} \tag{10-11}$$

The constraints for the edge is derived by substituting the parameter t from one of the scalar equations in Equation 10-10 into the remaining ones. In 2D the constraint equations become

$$(x+u-x_{d1})a_y-(y+v-y_{d1})a_x \tag{10-12}$$

In 3D the two constraints equations become

$$\begin{aligned}&(x+u-x_{d1})a_z-(z+w-z_{d1})a_x\\&(y+v-y_{d1})a_z-(z+w-z_{d1})a_y\end{aligned} \tag{10-13}$$

To avoid problems when the edge is directed in one of the coordinate axes directions, a third constraint is added. This constraint is a linear combination of the two earlier constraints:

$$(y+v-y_{d1})a_x-(x+u-x_{d1})a_y \tag{10-14}$$

You need a linear constraint in order for the solution of the small displacement problem to become independent of the solver. The linear relation for the displacement is

$$\mathbf{u} = \frac{\mathbf{u}_1(x_{n2}-x_n)+\mathbf{u}_2(x_n-x_{n1})}{(x_{n2}-x_{n1})}+u_{ax}(\mathbf{x}_2-\mathbf{x}_1) \tag{10-15}$$

where u_{ax} is the axial displacement along the edge, and x_n are a linear parameter along the edge

$$x_n = \frac{x(x_2-x_1)+y(y_2-y_1)+z(z_2-z_1)}{\sqrt{(x_2-x_1)^2+(y_2-y_1)^2+(z_2-z_1)^2}} \tag{10-16}$$

Eliminating u_{ax} from Equation 10-15 results in the following linear constraint in 2D

$$\begin{aligned}&\left[\frac{u_1(x_{n2}-x_n)+u_2(x_n-x_{n1})}{(x_{n2}-x_{n1})}-u\right](y_2-y_1)-\\&\qquad\left[\frac{v_1(x_{n2}-x_n)+v_2(x_n-x_{n1})}{(x_{n2}-x_{n1})}-v\right](x_2-x_1)=0\end{aligned} \tag{10-17}$$

and the following three linear constraints in 3D:

$$
\begin{aligned}
&\left[\frac{u_1(x_{n2}-x_n)+u_2(x_n-x_{n1})}{(x_{n2}-x_{n1})}-u\right](z_2-z_1)- \\
&\qquad\left[\frac{w_1(x_{n2}-x_n)+w_2(p-x_{n1})}{(x_{n2}-x_{n1})}-w\right](x_2-x_1) = 0 \\
&\left[\frac{v_1(x_{n2}-x_n)+v_2(x_n-x_{n1})}{(x_{n2}-x_{n1})}-v\right](z_2-z_1)- \\
&\qquad\left[\frac{w_1(x_{n2}-x_n)+w_2(x_n-x_{n1})}{(x_{n2}-x_{n1})}-w\right](y_2-y_1) = 0 \\
&\left[\frac{v_1(x_{n2}-x_n)+v_2(x_n-x_{n1})}{(x_{n2}-x_{n1})}-v\right](x_2-x_1)- \\
&\qquad\left[\frac{u_1(x_{n2}-x_n)+u_2(x_n-x_{n1})}{(x_{n2}-x_{n1})}-u\right](y_2-y_1) = 0
\end{aligned}
\tag{10-18}
$$

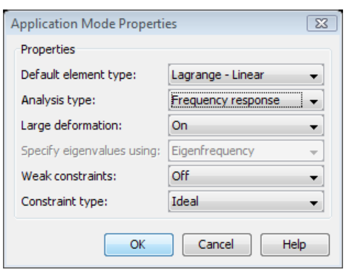
Application Mode Description

This section describes how to define a truss model. It is divided into the following sections:

- Properties
- Scalar Variables
- Material
- Cross Section
- Constraint
- Load
- Thermal Coupling
- Initial Stress and Strain

Properties

To open the **Application Mode Properties** dialog box, choose **Properties** from the **Physics** menu.



In the **Application Mode Properties** dialog box you control different global settings for the model.

- **Default element type**: The selected finite element type that makes up the discretized finite element model is the default on all new boundaries/edges, and the choice does not affect boundaries/edges already created. Available elements are:
  - **Lagrange - Linear**
  - **Lagrange - Quadratic**
  - **Lagrange - Cubic**
  - **Lagrange - Quartic**
  - **Lagrange - Quintic**
- **Analysis type**: A list of different analyses to perform. It affects both the equations and what solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. The available analysis types use the following solvers:

ANALYSIS TYPE	COMSOL MULTIPHYSICS SOLVER
Static	Stationary
Eigenfrequency	Eigenvalue
Time dependent (Transient)	Time dependent
Frequency response	Parametric
Parametric	Parametric
Quasi-static transient	Time dependent
Linear Buckling	Eigenvalue

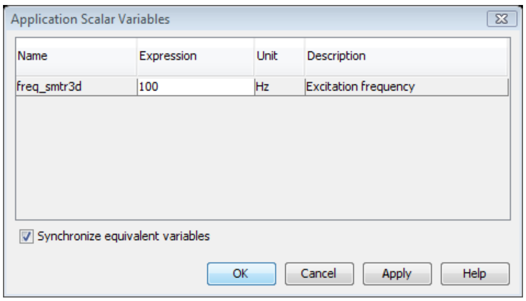
- **Large deformation**: This list controls whether or not the model should support large deformations.
- **Specify eigenvalues using**: This list controls how to work with eigenmode analyses. Here you should specify **Eigenvalue** or **Eigenfrequency/Critical load factor**; this property is enabled only for eigenfrequency and linear-buckling analyses.
- **Weak constraints**: Controls whether or not weak constraints are active. Use weak constraints for accurate reaction-force computation. When weak constraints are enabled, all constraints are weak by default, but it is possible to change this setting for individual domains.
- **Constraint type**: Constraints can be ideal or nonideal (see “Ideal vs. Non-Ideal Constraints” on page 301 in the *COMSOL Multiphysics Modeling Guide*).

Scalar Variables

There are two different scalar variables:

- Excitation frequency, `freq`, which is applicable only for frequency response analysis.
- Complex angular frequency, `jomega`, which is applicable only for eigenfrequency analysis. You normally do not need to edit the complex angular frequency.

The **Scalar Variables** menu item on the **Physics** menu is enabled only when **Frequency response**, **Damped eigenfrequency**, or **Eigenfrequency** is selected as **Analysis type** in the **Application Mode Properties** dialog box.

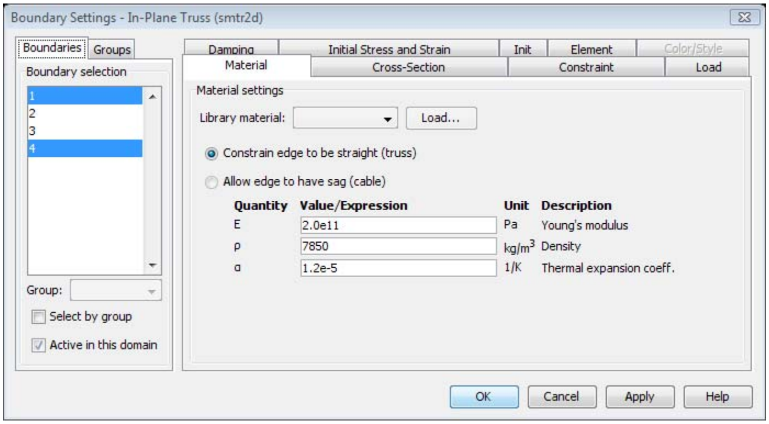


The excitation frequency is the frequency of the harmonic loads/constraints in a frequency response analysis.

When **Frequency response** is selected as analysis type, the default solver is the parametric solver, making it easy to perform a frequency sweep over several excitation frequencies in a single analysis. In this case, enter `freq_smtr2d` as the **Parameter name** on the **Parametric** page in the **Solver Parameters** dialog box. Doing so makes the values entered in the **Parameter values** edit field override the excitation frequency entered in the **Application Scalar Variables** dialog box.

Material

The material properties are defined on the **Material** page in the **Boundary Settings** dialog box for the In-Plane Truss and in the **Edge Settings** dialog box for the 3D Truss.



The material properties are shown in the table below.

PARAMETER	VARIABLE	DESCRIPTION
E	E	Young's modulus
ρ	rho	Density
α	alpha	Thermal expansion coefficient

Young's modulus Defines the modulus of elasticity, E of the material. It is the spring stiffness in Hooke's law, shown below in 1D form

$$\sigma = E\varepsilon$$

Density This material property, ρ, specifies the density of the material.

Thermal expansion coefficient Defines how much a material expands due to an increase in temperature.

$$\varepsilon_{th} = \alpha(T - T_{ref})$$

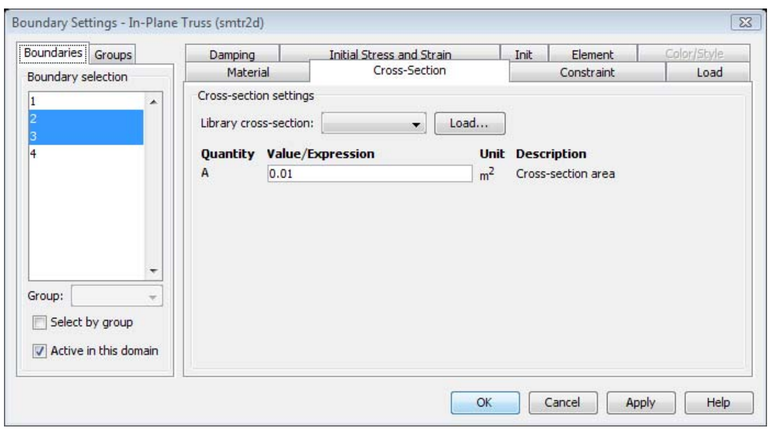
where ε_{th} is the thermal strain, T is the strain temperature and T_{ref} is the stress free reference temperature.

The **Constrain edge to be straight (truss)** and **Allow edge to have sag (cable)** buttons control the addition of an additional constraint, forcing the edge to be straight. The default is to add the constraint. Using this additional constraint removes the need to use a mesh with only one element per edge. The problem with internal nodes is that they makes the problem singular because the truss only has stiffness in the axial direction. The same applies when using higher-order elements. The additional constraint increases the solution time, especially for large 3D and transient problems. The remedy to this is to turn off the constraint option (click the **Allow edge to have sag (cable)** button) and use linear elements together with a very coarse mesh consisting of only one element/edge.

For problems where you want to model the sag and do not have a straight line between the edge points, click the **Allow edge to have sag (cable)** button and use that setting together with the **Large deformation** option and a suitable mesh with internal nodes.

Cross-Section Properties

You define cross-sectional properties on the **Cross Section** page in the **Edge Settings/ Boundary Settings** dialog box.



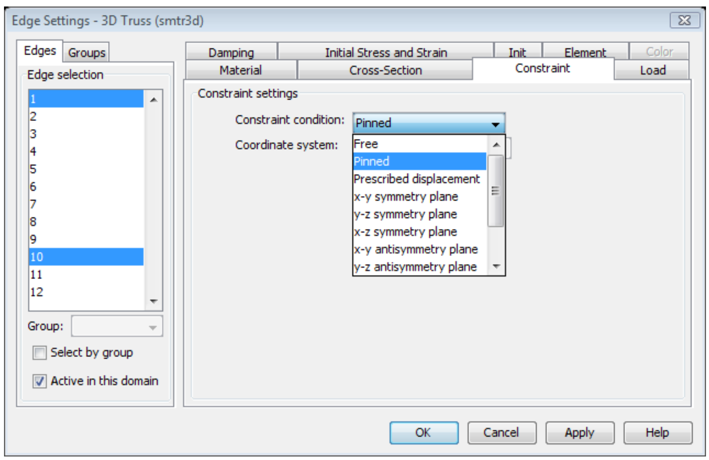
The only cross-section property in these application modes is the cross-section area:

PARAMETER	VARIABLE	DESCRIPTION	COMMENT
A	A	Cross-section area	

Constraints

A constraint specifies the displacement of a certain part of the truss. Constraints can be defined on all valid domain levels such as points and edges/boundaries. You control the constraints from the **Constraint** page in the **Boundary Settings**, **Edge Settings**, and **Point Settings** dialog boxes.

The following figure shows the **Boundary Settings** dialog box for the 3D Truss application mode, but the page looks similar on all domain levels in both truss application modes.



An example of a truss Constraint page, taken here from the 3D Truss application mode Edge Settings dialog box.

Within the dialog box, use the **Constraint condition** list to specify the type of constraint. You can choose from the following options:

CONSTRAINT CONDITION	POINT	BOUNDARY/ EDGE	USE WHEN
Free	√	√	The domain has no constraint
Pinned	√	√	The displacement in the domain is fixed in all directions
Roller		√ (2D only)	The normal displacement is constrained
Prescribed displacement	√	√	The displacement in any direction need to be prescribed
Symmetry plane		√ (2D only)	The boundary is a symmetry plane

CONSTRAINT CONDITION	POINT	BOUNDARY/ EDGE	USE WHEN
x-y symmetry plane	√		The selected coordinate system's xy-plane is a symmetry plane
y-z symmetry plane	√		The selected coordinate system's yz-plane is a symmetry plane
x-z symmetry plane	√		The selected coordinate system's xz-plane is a symmetry plane
Antisymmetry plane		√ (2D only)	The boundary is an antisymmetry plane
x-y antisymmetry plane	√		The selected coordinate system's xy-plane is an antisymmetry plane
y-z antisymmetry plane	√		The selected coordinate system's yz-plane is an antisymmetry plane
x-z antisymmetry plane	√		The selected coordinate system's xz-plane is an antisymmetry plane
Prescribed velocity	√	√	The velocity in any direction need to be prescribed (only available for frequency response analysis)
Prescribed acceleration	√	√	The acceleration in any direction need to be prescribed (only available for frequency response analysis)

The symmetry or antisymmetry condition has the following interpretation.

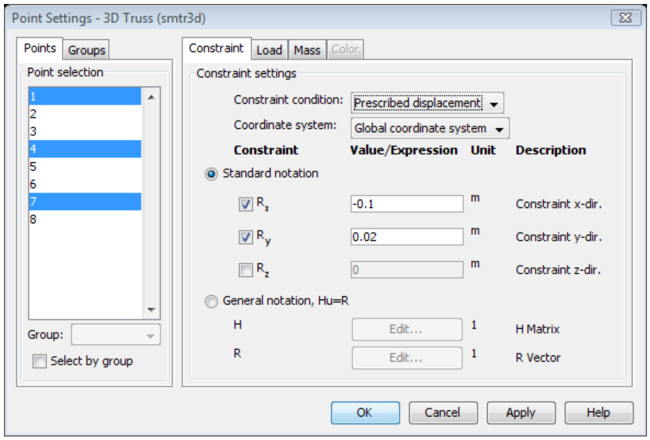
CONDITION	X-DISPLACEMENT	Y-DISPLACEMENT	Z-DISPLACEMENT
x-y symmetry plane			√
y-z symmetry plane	√		
x-z symmetry plane		√	
x-y antisymmetry plane	√	√	
y-z antisymmetry plane		√	√
x-z antisymmetry plane	√		√

The **Coordinate system** list lets you control in which coordinate system you want the constraint defined. Available options are:

- Global coordinate system
- Tangential and normal coordinate system, only available on boundaries for the in-plane truss.

- User-defined coordinate systems, if there are any local coordinate systems defined. Read more about creation of coordinate system in the section “Coordinate Systems” on page 144.

When you select **Prescribed displacement** a number of new options appears in the dialog box and the **Constraint** page takes on this appearance:



The Constraint page showing the Prescribed displacement options.

You can prescribe a constraint in two ways:

- In standard notation you constrain each displacement direction independently. The check box in front of $\mathbf{R_x}$, $\mathbf{R_y}$, and $\mathbf{R_z}$ activates the constraint, and you can then enter the value or expression for the displacement in the corresponding edit fields. The default value is `0` (no displacement).
- In general notation, the H matrix and R vector in the relation

$$Hu = R$$

  make it possible to specify constraints as any linear combination of the available variables.

  For the In-Plane Truss application mode the relation is

$$H\begin{bmatrix} u \\ v \end{bmatrix} = R$$

  For the 3D Truss application mode the relation is

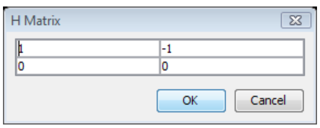
$$H\begin{bmatrix}u\\v\\w\end{bmatrix} = R$$

To enter the H matrix and the R vector, use special matrix dialog boxes that you open by clicking the corresponding **Edit** buttons. For example, you can achieve the condition $u = v$ in the In-Plane Truss application mode using the settings

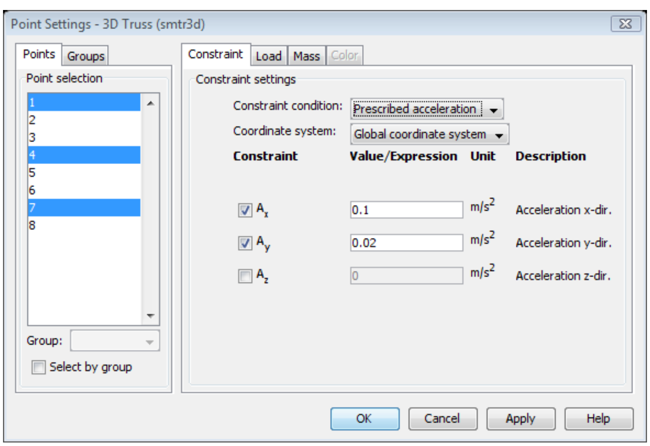
$$H = \begin{bmatrix}1 & -1\\0 & 0\end{bmatrix} \qquad R = \begin{bmatrix}0\\0\end{bmatrix},$$

which force the domain to move only diagonally in the x-y plane.

The **H Matrix** dialog box for the above example is



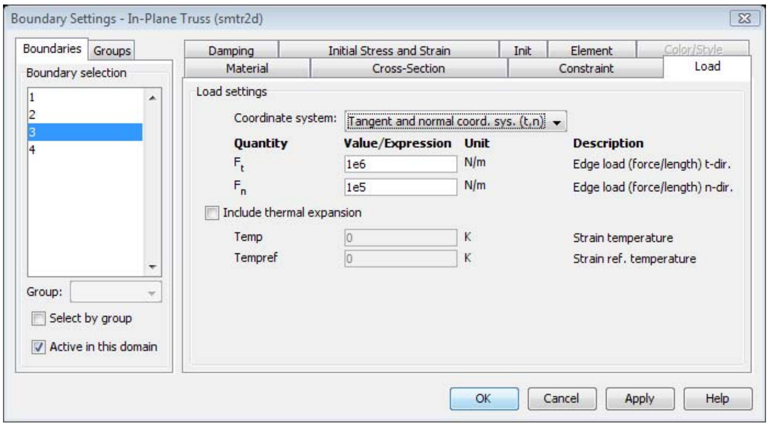
In a frequency response analysis you have the possibility to specify not only a harmonic displacement but also a harmonic velocity or acceleration. You specify your **Prescribed velocity** and **Prescribed acceleration** in the same way as **Prescribed displacement** using **Standard notation**.



Constraint page showing the Prescribed acceleration settings.

Loads

A load is a general name for all forces applied to the structure. You can specify loads on all domain types using the **Load** page in the **Boundary Settings**, **Edge Settings**, and **Point Settings** dialog boxes. The following picture shows the **Boundary Settings** dialog box for the In-Plane Truss application mode, but the page looks similar on all domain levels.



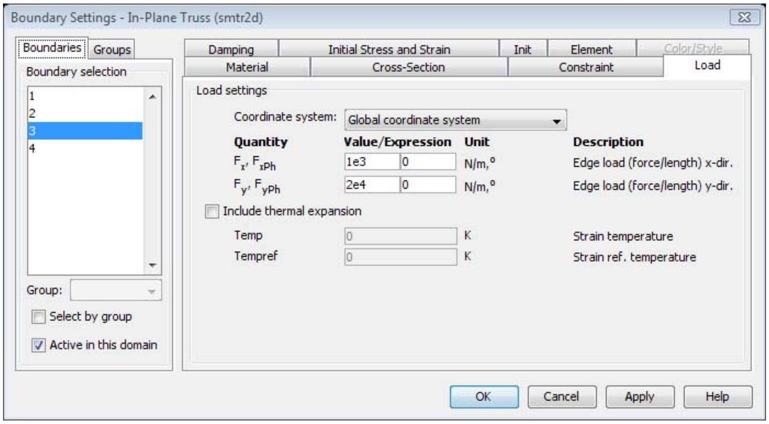
The loads are defined in the following way. The SI unit is shown in parenthesis.

POINT	EDGE, BOUNDARY
force (N)	force/length (N/m)

With the **Coordinate system** list you control in what coordinate system the load is defined. Available options are:

- Global coordinate system
- Tangential and normal coordinate system, only available on boundaries for the in-plane truss.
- User-defined coordinate systems, if there are any local coordinate systems defined. Read more about creation of coordinate system in the section "Coordinate Systems" on page 144.

For the frequency response analysis type, you need to specify additional input data. The analysis type is controlled from the **Application Mode Properties** dialog box. When frequency response is selected as analysis type, the **Load** page changes appearance to



For frequency response analysis the harmonic load is split into 3 different parameters:

- the amplitude value, F
- the amplitude factor, F_{Amp} (a dimensionless number; the default value is 1)
- the phase (F_{Ph})

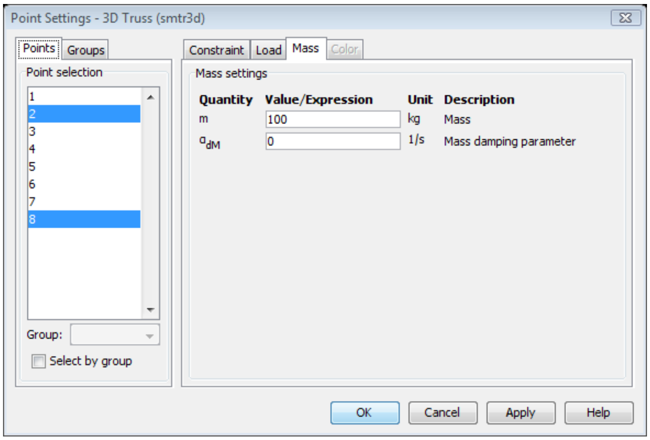
Together they define a harmonic load whose amplitude and phase shift can vary with the excitation frequency f.

$$F_{\mathrm{freq}} = F \cdot F_{\mathrm{Amp}}(f) \cdot \cos(2\pi f + F_{\mathrm{Ph}}(f))$$

On the edge and boundary domain level additional options are available controlling if and how thermal strains should be included in the analysis. They are explained in the section “Thermal Coupling” on page 295.

Discrete Mass

Discrete mass is concentrated to a point in contrast to distributed mass modeled through the density and area of the truss. You specify discrete mass on the **Mass** page in the **Point Settings** dialog box.



The mass properties are shown in the table below.

PARAMETER	VARIABLE	DESCRIPTION	SI UNITS
m	m	Mass	kg
α_{dM}	alphadM	Mass damping parameter	1/s

Thermal Coupling

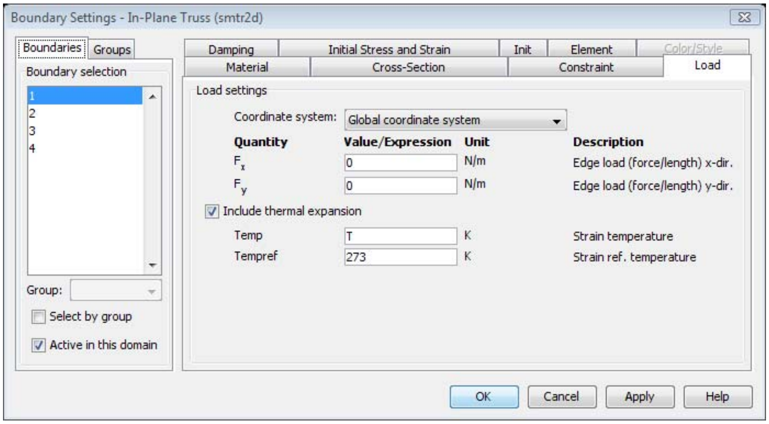
Material expands with temperature, which causes thermal strains to develop in the material. The trusses can handle any temperature variation along the truss. The thermal strains together with the initial strains and elastic strains from structural loads form the total strain.

$$\varepsilon = \varepsilon_{\mathrm{el}} + \varepsilon_{\mathrm{th}} + \varepsilon_i$$

where

$$\varepsilon_{\mathrm{th}} = \alpha(T - T_{\mathrm{ref}})$$

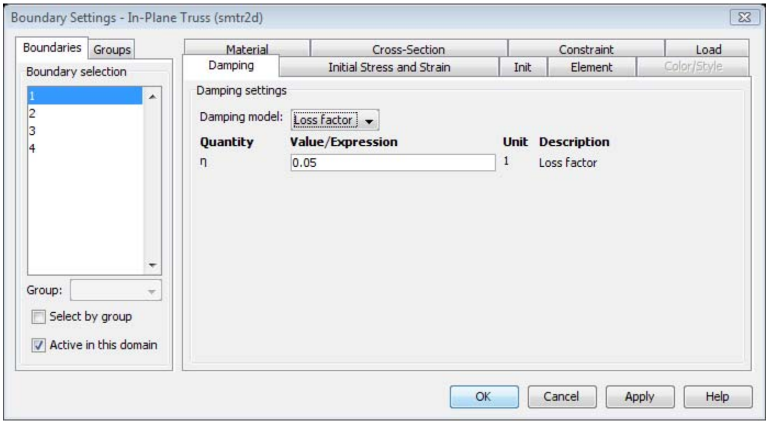
Thermal coupling means that the thermal expansion is included in the analysis. Details on thermal coupling is found on page 281. Thermal effects are specified on the **Load** page in the **Edge Settings** or **Boundary Settings** dialog box.



The **Include thermal expansion** check box adds thermal effects. In the **Temp** and **Tempref** edit fields you specify the strain temperature T and stress free reference temperature T_{ref}, respectively. Use the **Material** page to define the thermal expansion coefficient (described in "Material" on page 287). T and T_{ref} can be any expression and can be a dependent variable for temperature from another application modes solving the heat transfer problem. The temperature coupling can be used in any type of analysis.

Damping

In transient and frequency response analyses you have the possibility to model undamped or damped problems. In the Structural Mechanics Module you can specify damping on the subdomain level using the **Damping** page that appears in the **Boundary Settings** (2D) or **Edge Settings** (3D) dialog box. From the **Damping models** list you can select **No damping**, **Rayleigh**, or **Loss factor**, and the layout of the dialog box changes for each model.



Damping page when Rayleigh damping is selected.

Note: Loss factor damping is valid only for frequency response analysis. If you choose transient analysis and loss factor damping, COMSOL Multiphysics solves the model with no damping.

Table 10-1 and the following text describe the parameters that define damping:

TABLE 10-1: PARAMETERS FOR DAMPING MODELS

PARAMETER	VARIABLE	DESCRIPTION	DAMPING MODEL
α_{dM}	`alphadM`	Mass-damping parameter	Rayleigh
β_{dK}	`betadK`	Stiffness-damping parameter	Rayleigh
η	`eta`	Loss factor	Loss factor

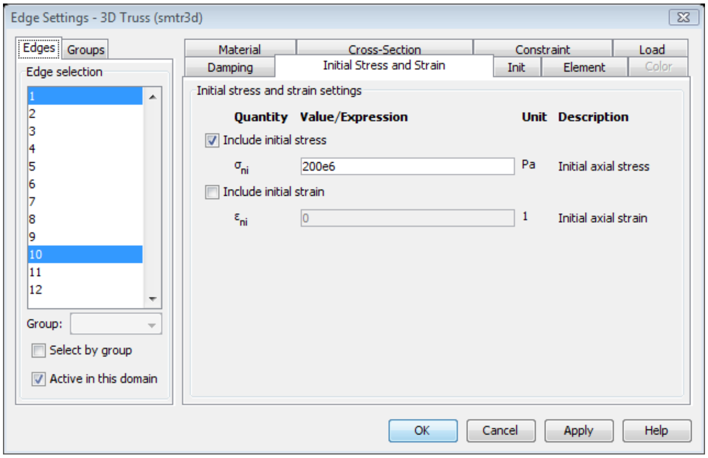
Mass damping parameter Defines the Rayleigh damping model’s mass damping, α_{dM}.

Stiffness damping parameter Defines the Rayleigh damping model’s stiffness damping, β_{dK}.

Loss factor Defines the loss factor η for the loss factor damping model.

Initial Stress and Strain

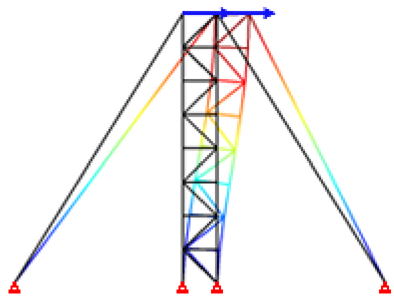
The analysis can include initial stress and strain. You specify the initial stress and strain on the **Initial Stress and Strain** page in the **Edge Settings** or **Boundary Settings** dialog box.



It is possible to control the options to include initial stress and strain independently using the **Include initial stress** and **Include initial strain** check boxes.

In-Plane Truss Application Mode

Use the In-Plane Truss application mode to analyze planar lattice trusses or sagging cable-like structures.



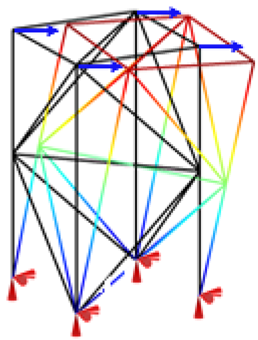
In-Plane Truss application modes are defined on edges in 2D. All settings for the application mode are described in “Application Mode Description” on page 284.

Variables and Space Dimensions

The degrees of freedom (dependent variables) are the global displacements u and v in the global x and y directions, respectively.

3D Truss Application Mode

Use the 3D Truss application mode to model three-dimensional trusses or sagging cable-like structures.



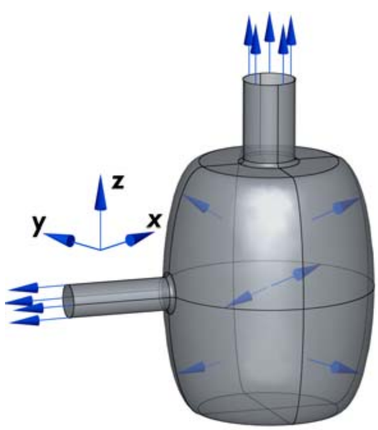
3D Truss application modes are defined on edges in 3D. All settings for the application mode appear in "Application Mode Description" on page 284.

Variables and Space Dimensions

The degrees of freedom (dependent variables) are the global displacements u, v, and w in the global x, y, and z directions, respectively.

11

Shells



A shell is a thin-walled structure in 3D where you can assume a simple form for the displacement's variation through the thickness. Using this approximation, it is possible to develop a model for the deformation that is closer to the 2D plane stress and Mindlin plate application modes than to the 3D solid. For this to give accurate results it is important that the structure can really be described as thin-walled.

When modeling using shells it is important to remember that the faces should be defined in the midplane of the real geometry.

A Shell application mode can be active either on free surfaces embedded in 3D or on the boundary of a solid 3D object. In the latter case, it can be used to model a reinforcement that stiffens the surface of a 3D solid.

The shell is described by its thickness and the material properties E, ρ, ν, α_{dM}, and β_{dK}. All properties are evaluated as constant within any mesh element but can vary from one element to the next.

The element used for the shell application mode is of Mindlin-Reissner type, which means that transverse shear deformation is accounted for. Because the element is a flat-faceted triangle, the membrane and bending actions are uncoupled. The membrane action is modeled by a constant-strain triangle with true drilling rotations (Allman triangle, D. J. Allman; see Ref. 1). The bending action is modeled by the bending part of an Argyris TRIC triangle element; see J. Argyris et al. (Ref. 2) and C. Pacoste (Ref. 3) for further details.

Note: The shell application mode requires a triangular mesh and will not work with a quadrilateral mesh.

The dependent variables are the displacements u, v, and w in the global x, y, and z directions, and the rotations θ_x, θ_y, and θ_z about the global coordinate axes. The degrees of freedom defined by the shell element correspond to the values of the dependent variables in the three triangle vertices.

In contrast to the rest of the Structural Mechanics Module, the Shell application mode contains a mixture between a user-modifiable variational equation and a low-level element. The stiffness and mass matrices are assembled directly by the low-level shell element class (see the documentation of `elshell_arg2` in the *COMSOL Multiphysics Reference Guide* for details), but the constraints and loads are assembled by linear Lagrange elements. Therefore the Shell application mode has somewhat limited multiphysics capabilities: The presence of dependent variables in expressions for the material properties are not accounted for in the Jacobian. It is possible, however, to use the dependent variables of another application mode in the loads on the shell.

References

1. D.J. Allman: “Evaluation of the constant strain triangle with drilling rotations,” *Int. J. Numer. Meth. Eng.*, vol. 26, pp. 2645–2655, 1988.

2. J. Argyris, L. Tenek, and L. Olofsson: “TRIC: a simple but sophisticated 3-node triangular element based on 6 rigid-body and 12 straining modes for fast computational simulations of arbitrary isotropic and laminated composite shells,” *Comput. Methods Appl. Mech. Engrg.*, vol. 145, p. 11–85, 1997.

3. C. Pacoste: *A flat facet three node element for shell analysis—some theoretical and numerical aspects*, Royal Institute of Technology, Department of Structural Engineering, Technical report 1999:20, Structural Mechanics, 1999.

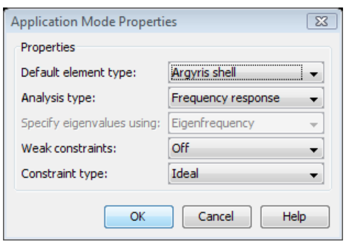
Application Mode Description

This section describes how to define a shell model. It is divided into the following sections:

- Properties
- Scalar Variables
- Material
- Constraint
- Load
- Postprocessing

Properties

The **Application Mode Properties** dialog box is opened from the **Physics** menu.



In the **Application Mode Properties** dialog box you control different global settings for the model.

- **Analysis type**: A list of different analyses to perform. It affects both the equations and what solver to use through the **Auto select solver** option in the **Solver Parameters** dialog box. The available analysis types use the following solvers.

ANALYSIS TYPE	COMSOL MULTIPHYSICS SOLVER
Static	Stationary
Eigenfrequency	Eigenvalue
Time dependent	Time dependent
Frequency response	Parametric
Parametric	Parametric
Quasi-static transient	Time dependent

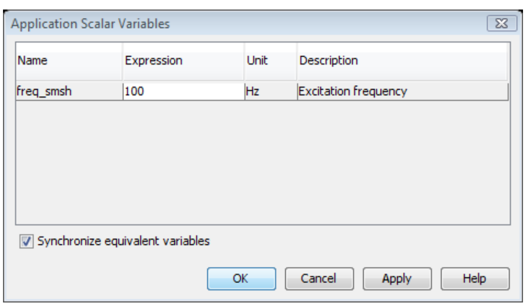
- **Specify eigenvalues using**: Controls if eigenvalues or eigenfrequencies should be used when specifying parameters for the eigenvalue solver and if the result is returned as eigenvalues or eigenfrequencies.
- **Weak constraints**: Controls whether or not weak constraints are active. Use weak constraints for accurate reaction-force computation. When weak constraints are enabled, all constraints are weak by default, but it is possible to change this setting for individual domains.
- **Constraint type**: Constraints can be ideal or nonideal (see “Ideal vs. Non-Ideal Constraints” on page 301 in the *COMSOL Multiphysics Modeling Guide*).

Scalar Variables

There are two different scalar variables:

- Excitation frequency, `freq`, which is applicable only for frequency response analysis.
- Complex angular frequency, `jomega`, which is applicable only for eigenfrequency analysis. You normally do not need to edit the complex angular frequency.

The **Scalar Variables** menu item on the **Physics** menu is enabled only when **Frequency Response, Damped Eigenfrequency**, or **Eigenfrequency** is selected as **Analysis type** in the **Application Mode Properties** dialog box.

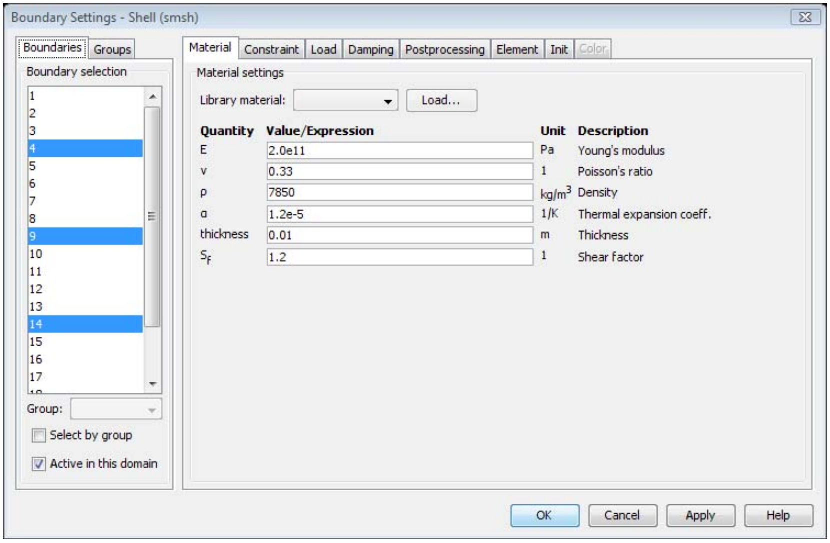


The excitation frequency is the frequency of the harmonic loads in a frequency response analysis.

When **Frequency response** is selected as the analysis type, the default solver is the parametric solver making it easy to perform a frequency sweep over several excitation frequencies in a single analysis. In this case `freq_smsh` is entered as the **Parameter name** on the **Parametric** page in the **Solver Parameters** dialog box and the values entered in the **Parameter values** edit field override the excitation frequency entered in the **Application Scalar Variables** dialog box.

Material

The material properties are defined on the **Material** page in the **Boundary Settings** dialog box.



The material properties are shown in the table below.

PARAMETER	VARIABLE	DESCRIPTION
E	E	Young's modulus
ν	nu	Poisson's ratio
S_f	Sf	Shear factor
ρ	rho	Density
α	alpha	Thermal expansion coefficient
th	thickness	Thickness
α_{dM}	alphadM	Mass damping parameter
β_{dK}	betadK	Stiffness damping parameter

Young's modulus Defines the modulus of elasticity, E of the material. For an isotropic material, it is the spring stiffness in Hooke's law, shown below in 1D form

$$\sigma = E\varepsilon$$

where σ is the stress and ε is the strain.

Poisson's ratio Denoted by ν, defines the normal strain in the perpendicular direction, generated from a normal strain in the other direction.

$$\varepsilon_{\perp} = -\upsilon\varepsilon_{||}$$

Shear Factor Denoted by S_f, the shear factor affects the shear stiffness. For a homogeneous material, $S_f = 1.2$.

Density This material property, ρ, specifies the density of the material.

Thermal expansion coefficient Defines how much a material expands due to an increase in temperature:

$$\varepsilon_{\mathrm{th}} = \alpha(T - T_{\mathrm{ref}})$$

where $\varepsilon_{\mathrm{th}}$ is the thermal strain and α is the thermal expansion coefficient. The thermal expansion coefficient models thermal strain in the material.

Thickness Defines the thickness of the shell.

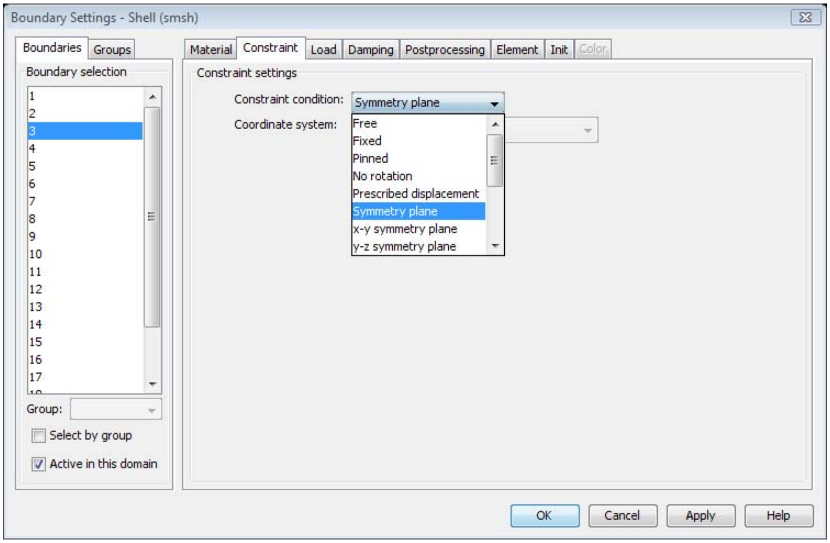
Mass damping parameter Defines the Rayleigh damping models mass damping, α_{dM}.

Stiffness damping parameter Defines the Rayleigh damping models stiffness damping, β_{dK}.

Constraint

A constraint specifies the displacements and rotations of certain parts of a shell. Constraints can be defined on all domain levels such as points, edges, and faces. The constraint is controlled from the **Constraint** page in the **Boundary/Edge/Point Settings** dialog boxes.

Below is the **Boundary Settings** dialog box.



The Constraint page from the Boundary Settings dialog box.

Within the dialog box the **Constraint condition** list lets you control what type of constraint you want to define. You have the following options to choose between:

CONSTRAINT CONDITION	EDGE	BOUNDARY	USE WHEN
Free	√	√	The domain has no constraint
Pinned	√	√	The displacement in the domain is fixed in all directions
Fixed	√	√	The displacement and rotations in the domain are fixed in all directions
No rotation	√	√	The rotations in the domain are fixed in all directions
Prescribed displacement	√	√	The displacement or rotation in any direction need to be prescribed
Symmetry plane		√	The boundary is a symmetry plane
x-y symmetry plane	√	√	The selected coordinate system’s x-y plane is a symmetry plane
y-z symmetry plane	√	√	The selected coordinate system’s y-z plane is a symmetry plane
x-z symmetry plane	√	√	The selected coordinate system’s x-z plane is a symmetry plane

CONSTRAINT CONDITION	EDGE	BOUNDARY	USE WHEN
Antisymmetry plane		√	The boundary is an antisymmetry plane
x-y antisymmetry plane	√	√	The selected coordinate system's x-y plane is an antisymmetry plane
y-z antisymmetry plane	√	√	The selected coordinate system's y-z plane is an antisymmetry plane
x-z antisymmetry plane	√	√	The selected coordinate system's x-z plane is an antisymmetry plane
Prescribed velocity	√	√	The velocity and angular velocity in any direction need to be prescribed, only available for frequency response analysis
Prescribed acceleration	√	√	The acceleration or angular acceleration in any direction need to be prescribed, only available for frequency response analysis

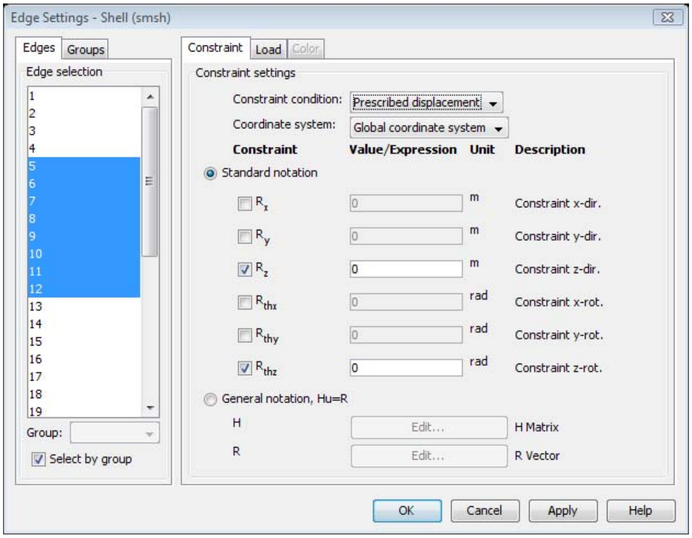
The symmetry or antisymmetry condition has the following interpretation.

CONDITION	X-DISP	Y-DISP	Z-DISP	X-ROT	Y-ROT	Z-ROT
x-y symmetry plane			√	√	√	
y-z symmetry plane	√				√	√
x-z symmetry plane		√		√		√
x-y antisymmetry plane	√	√				√
y-z antisymmetry plane		√	√	√		
x-z antisymmetry plane	√		√		√	

With the **Coordinate system** list you control in what coordinate system the constraint is defined. Available options are:

- Tangential and normal coordinate system, only applicable on faces.
- Shell local coordinate system, only applicable on faces.
- Global coordinate system.
- User-defined coordinate systems, if there are any local coordinate systems defined. Read more about creation of coordinate system in the coordinate system section.

When you select **Prescribed displacement** a number of new option appears in the dialog box and the **Constraint** page takes on this appearance:



The Constraint page showing the Prescribed displacement options.

The constraint can be described using standard or general notation. This is controlled with the **Standard notation** button and the **General notation, Hu=R** button.

In standard notation you constrain the displacement and rotations independently. The check box in front of $\mathbf{R_x}$, $\mathbf{R_y}$, $\mathbf{R_z}$, $\mathbf{R_{thx}}$, $\mathbf{R_{thy}}$, and $\mathbf{R_{thz}}$ activates the constraint, the value/expression of the displacement can then be entered in the edit fields. The default value is 0.

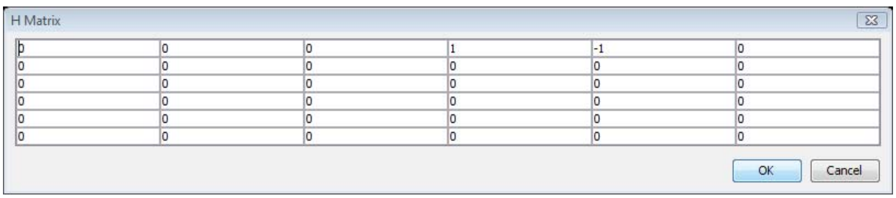
In general notation, the H matrix and the R vector, related by the equation

$$H\begin{bmatrix} u \\ v \\ w \\ \theta_x \\ \theta_y \\ \theta_z \end{bmatrix} = R$$

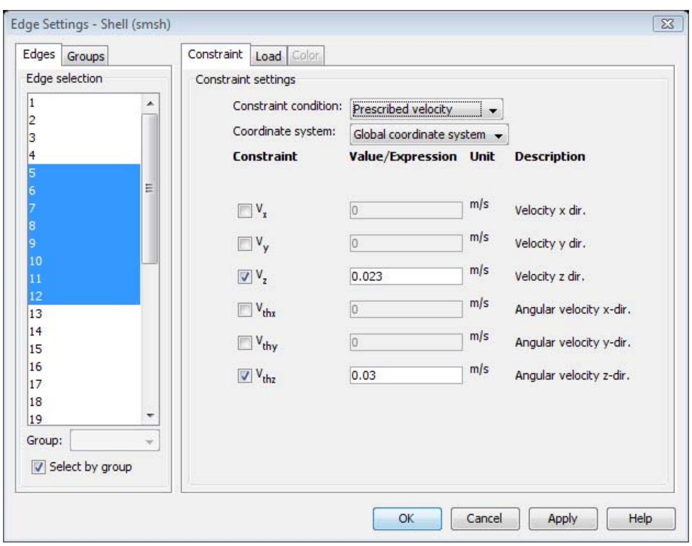
let you specify constraints as any linear combination of displacement and rotation components. You enter the H matrix and the R vector in special matrix dialog boxes by clicking the corresponding **Edit** buttons. For example the condition $\theta_x = \theta_y$ can be achieved using the settings

$$H = \begin{bmatrix} 0 & 0 & 0 & 1 & -1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & & 0 & 0 & 0 \end{bmatrix}, \qquad R = \begin{bmatrix} 0 \\ 0 \end{bmatrix}$$

The **H Matrix** dialog box for the above example is



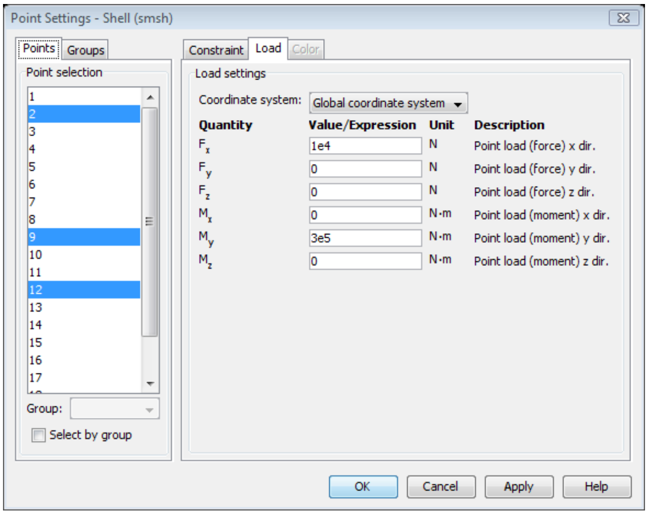
In a frequency response analysis you have the possibility to specify not only a harmonic displacement and rotation but also a harmonic velocity/angular velocity or acceleration/angular acceleration. You specify your **Prescribed velocity** and **Prescribed acceleration** in the same way as **Prescribed displacement** using **Standard notation**.



Constraint page showing the Prescribed velocity settings.

Loads

Load is a general name for forces and moments applied to the structure. You can specify loads on all domain types. To do so, click the **Load** tab in the **Boundary Settings**, **Edge Settings**, and **Point Settings** dialog boxes. The following picture shows the **Point Settings** dialog box, but the page looks similar on all domain levels.



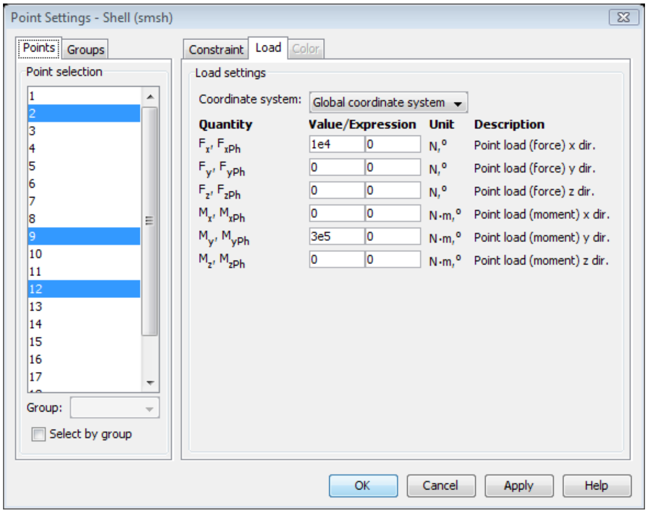
In the **Boundary Settings** and **Edge Settings** dialog boxes you have an option to specify the load in different ways using the thickness. The loads can be defined on different domains in the following way. The SI unit is shown in parenthesis.

POINT	EDGE	BOUNDARY/FACE
force (N), moment (Nm)	force/area (N/m^2), moment/area (N/m) or force/length (N/m), moment/length (N)	force/volume (N/m^3), moment/volume (N/m^2) or force/area (N/m^2), moment/area (N/m)

With the **Coordinate system** list you control in what coordinate system the load is defined. Available options are:

- Tangential and normal coordinate system, only applicable on faces.
- Shell local coordinate system, only applicable on faces.
- Global coordinate system.
- User-defined coordinate systems, if there are any local coordinate systems defined. Read more about creation of coordinate system in the coordinate system section.

For the frequency response analysis type, additional input is specified. The analysis type is controlled from the **Application Mode Properties** dialog box. When frequency response is selected as analysis type, the **Load** page changes appearance:



For frequency response analysis the harmonic load is split in three different parameters:

- The amplitude value, F
- The amplitude factor, F_{Amp} (a dimensionless number; the default value is 1)
- The phase (F_{Ph}).

Together they define a harmonic load whose amplitude and phase shift can vary with the excitation frequency f.

$$F_{\mathrm{freq}} = F \cdot F_{\mathrm{Amp}}(f) \cdot \cos(2\pi f + F_{\mathrm{Ph}}(f))$$

Thermal Coupling

Material expands with temperature, which causes thermal strains to develop in the material. The thermal strains together with elastic strains from structural loads form the total strain.

$$\varepsilon = \varepsilon_{\mathrm{el}} + \varepsilon_{\mathrm{th}}$$

where

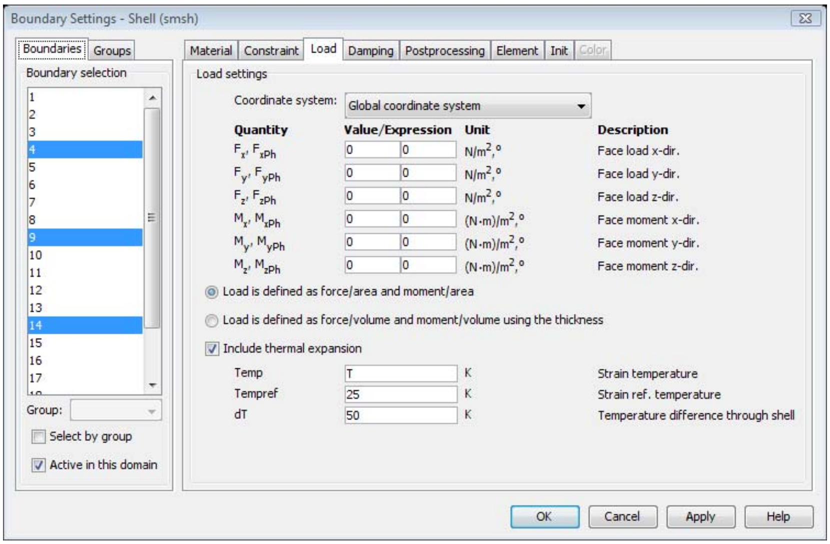
$$\varepsilon_{\mathrm{th}} = \alpha(T - T_{\mathrm{ref}})$$

Thermal coupling means that the thermal expansion is included in the analysis.

The temperature is assumed to vary linearly through the thickness of the shell.

$$T = T_0 + \Delta T \frac{z_l}{\mathrm{th}}$$

Thermal effects are specified on the **Load** page in the **Subdomain Settings** dialog box.

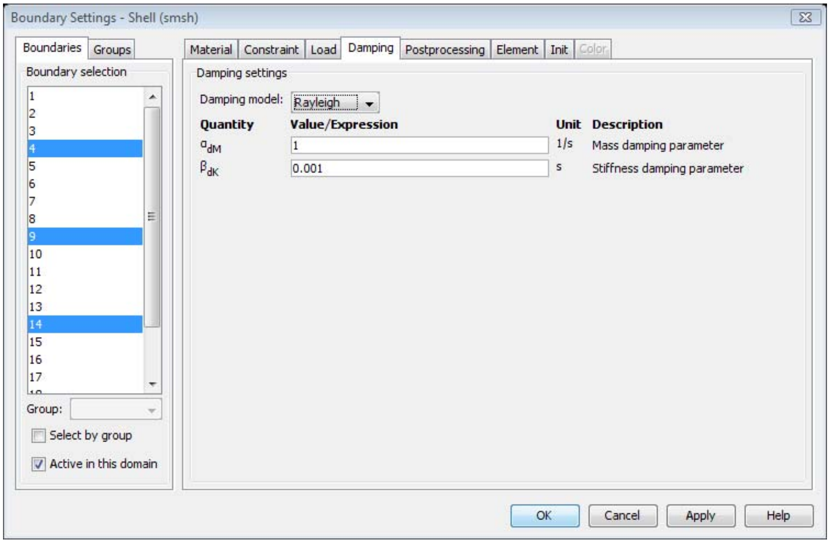


The **Include thermal expansion** check box adds thermal effects. In the **Temp**, **Tempref**, and **dT** edit fields the strain temperature T, reference temperature T_{ref}, and temperature difference through the shell dT are specified. The thermal expansion coefficient are specified on the **Material** page described in the Material section on page 306. T, T_{ref}, and dT can be any expression and are typically another variable solved for in a heat transfer application mode. The temperature coupling can be used in any type of analysis.

Damping

In transient and frequency response analyses you have the possibility to model undamped or damped problems. In the Structural Mechanics Module you can specify damping on the subdomain level using the **Damping** page that appears in the **Boundary**

Settings dialog box. From the **Damping models** list you can select **No damping**, **Rayleigh**, or **Loss factor**, and the layout of the dialog box changes for each model.



Damping page when Rayleigh damping is selected.

Note: Loss factor damping is valid only for frequency response analysis. If you choose transient analysis and loss factor damping, COMSOL Multiphysics solves the model with no damping.

Table 11-1 and the following text describe the parameters that define damping:

TABLE 11-1: PARAMETERS FOR DAMPING MODELS

PARAMETER	VARIABLE	DESCRIPTION	DAMPING MODEL
α_{dM}	alphadM	Mass-damping parameter	Rayleigh
β_{dK}	betadK	Stiffness-damping parameter	Rayleigh
η	eta	Loss factor	Loss factor

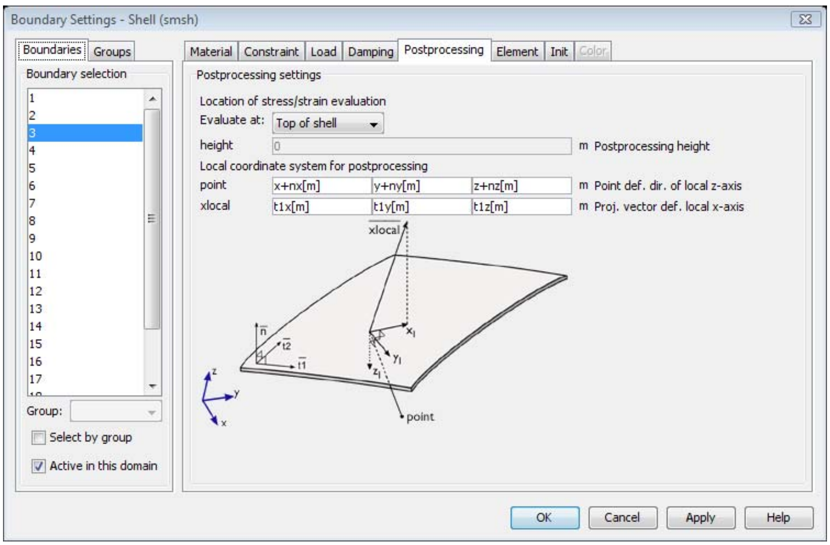
Mass damping parameter Defines the Rayleigh damping model's mass damping, α_{dM}.

Stiffness damping parameter Defines the Rayleigh damping model's stiffness damping, β_{dK}.

Loss factor Defines the loss factor η for the loss factor damping model.

Postprocessing

The predefined postprocessing variables include all nonzero stress and strain tensor components, principal stresses and strains, in-plane and out-of-plane forces, bending and torsional moments, and von Mises and Tresca effective stresses. The stress and strain tensor components and effective stresses can be evaluated at an arbitrary distance from the mid surface. This height is controlled from the **Postprocessing** page in the **Boundary Settings** dialog box.



With the **Evaluate at** list you control where the stress and strain should be evaluated, available options are:

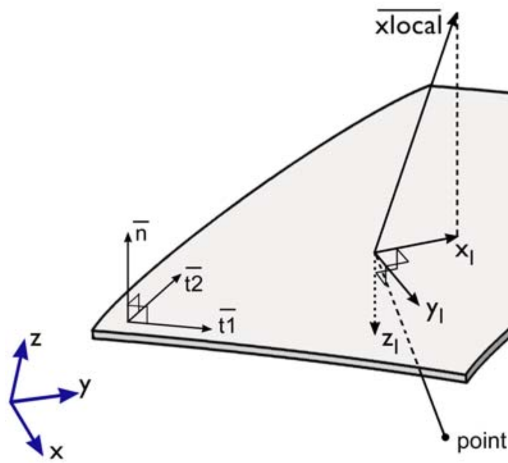
- Top of shell (default)
- Midplane of shell
- Bottom of shell
- Specified height

Select **Specified height** to specify a postprocessing height explicitly using the **height** edit field.

The displacements and rotations in radians and, for a transient analysis, the velocities and angular velocities can be plotted.

On the **Postprocessing** page, you can also specify a local coordinate system. The evaluation height, including the top and bottom of shell options, refer to the z axis direction in this coordinate system. In postprocessing mode some of the postprocessing variables can be plotted in the shell local coordinate system (that is, *not* the local coordinate system (t_1, t_2, n) defined by the geometry face object). The shell local system can be used to specify loads and constraints on faces. The local system is defined by a point and a vector in the following way:

The *point* defines the side of the shell to where the local z-axis is pointing. The local z-axis coincides either with the normal vector or with its mirror image with respect to the surface. See also the figure below.



A face geometry object has a unit normal vector $\mathbf{n}$ with the components n_x, n_y, and n_z. The default setting of the point uses these components and the independent variables x, y, and z so that the direction of the local z-axis coincides with n. For example the x coordinate's default setting is $n_x + x$.

The normal of a face geometry object can switch from one face to the next. The point is introduced as a means to specify the direction of the local z-axis irrespective of the details of the geometry representation. The point specifies the side where the local z-axis is positive.

The *vector* is used for defining the local x-axis. The vector is denoted x_{local} and the components of the vector are denoted $x_{\mathrm{local}x}$, $x_{\mathrm{local}y}$, and $x_{\mathrm{local}z}$. The x_{local} vector is projected onto the boundary surface. The projected vector x_l defines the direction of the local x-axis. The default settings use the geometry vector t_1 as x_{local} vector, for example the x component's default setting is t_{1x}.

12

Piezoelectric Application Modes

This chapter describes the application modes for modeling piezoelectric effects in the Structural Mechanics Module.

Theory Background

The Piezoelectric Effect

The piezoelectric effect manifests itself as a transfer of electric to mechanical energy and vice-versa. It is observable in many crystalline materials, while some materials such as quartz, Rochelle salt, and lead titanate zirconate ceramics display the phenomenon strongly enough for the phenomenon to be of practical use.

The *direct* piezoelectric effect consists of an electric polarization in a fixed direction when the piezoelectric crystal is deformed. The polarization is proportional to the deformation and causes an electric potential difference over the crystal.

The *inverse* piezoelectric effect, on the other hand, constitutes the opposite of the *direct* effect. This means that an applied potential difference induces a deformation of the crystal.

PIEZOELECTRIC CONVENTIONS

The documentation and the user interface use piezoelectric conventions as far as possible. These conventions differ from those used in other structural mechanics application modes. For instance, the numbering of the shear components in the stress-strain relation differs, as the following section describes. However, the names of the stress and strain components remain the same as in the other structural mechanics application modes.

Piezoelectric Constitutive Relations

It is possible to express the relation between the stress, strain, electric field, and electric displacement field in either a stress-charge or strain-charge form:

STRESS-CHARGE

$$\begin{aligned}\mathbf{T} &= c_E\mathbf{S} - e^T\mathbf{E}\\ \mathbf{D} &= e\mathbf{S} + \varepsilon_S\mathbf{E}\end{aligned}$$

STRAIN-CHARGE

$$\begin{aligned}\mathbf{S} &= s_E\mathbf{T} + d^T\mathbf{E}\\ \mathbf{D} &= d\mathbf{T} + \varepsilon_T\mathbf{E}\end{aligned}$$

The naming convention differs in piezoelectric theory compared to structural mechanics theory, but the piezoelectric application modes use the structural mechanics nomenclature. The strain is named ε instead of $\mathbf{S}$, and the stress is named σ instead of $\mathbf{T}$. This makes the names consistent with those used in the other structural mechanics application modes.

The numbering of the strain and stress components is also different in piezo and structural mechanics theory, and it is quite important to keep track of this aspect in order to give the correct material data. In structural mechanics the following is the most common numbering convention, and it is also the one used in the other structural mechanics application modes:

$$\sigma = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{bmatrix} \qquad \varepsilon = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ 2\varepsilon_{xy} \\ 2\varepsilon_{yz} \\ 2\varepsilon_{xz} \end{bmatrix}$$

In contrast, textbooks on piezoelectric effects and the IEEE standard on piezoelectric effects use the following numbering convention:

$$\sigma = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{yz} \\ \tau_{xz} \\ \tau_{xy} \end{bmatrix} \qquad \varepsilon = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{yz} \\ \gamma_{xz} \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ 2\varepsilon_{yz} \\ 2\varepsilon_{xz} \\ 2\varepsilon_{xy} \end{bmatrix}$$

The piezoelectric application modes employ the immediately preceding piezo numbering convention to make it easier to work with materials data and avoid mistakes.

The constitutive relation using COMSOL Multiphysics symbols for the different constitutive forms are thus:

STRESS-CHARGE

$$\sigma = c_E\varepsilon - e^T\mathbf{E}$$
$$\mathbf{D} = e\varepsilon + \varepsilon_0\varepsilon_{rS}\mathbf{E}$$

STRAIN-CHARGE

$$\varepsilon = s_E\sigma + d^T\mathbf{E}$$
$$\mathbf{D} = d\sigma + \varepsilon_0\varepsilon_{rT}\mathbf{E}$$

Most material data appears in the strain-charge form, and you can easily transform into the stress-charge form. COMSOL Multiphysics allows you to use both constitutive forms; simply select one, and COMSOL Multiphysics makes any necessary transformations. The following equations transform strain-charge material data to stress-charge data

$$c_E = s_E^{-1}$$
$$e = d\ s_E^{-1}$$
$$\varepsilon_S = \varepsilon_0\varepsilon_{rT} - d\ s_E^{-1}\ d^T$$

Material Models

In addition to modeling piezoelectric materials, the Piezoelectric application mode provides different material models for easier modeling of piezo components. This means, that in the subdomain settings of the application mode, you can define the material of each domain as:

- Piezoelectric
- Decoupled, isotropic
- Decoupled, anisotropic

The Piezoelectric material operates as described in the chapter above, whereas using the two other material models, you can model structural and electrical problems or either of them independently.

The structural part of the *Decoupled, isotropic* and *Decoupled, anisotropic* material operates as the linear elastic material with small deformations as described in “Continuum Application Modes” on page 159“Structural Mechanics Application

Modes” on page 59. However, the initial stress and strain and thermal expansion are not supported within the Piezoelectric application mode.

For the *Decoupled, isotropic* material you define the material using the Young’s modulus, E, and the Poisson ratio, ν. For the *Decoupled, anisotropic* material you define the full 6-by-6 elasticity matrix D. Note here, that you define D using the standard structural mechanics ordering. Thus the ordering of the D is different from the ordering of the piezoelectric c_E matrix.

Depending on the value of the *Electrostatics formulation* property (See “Electrical Formulations” below), the electrical part of the *Decoupled, isotropic* and *Decoupled, anisotropic* material solves either the electrostatics equation:

$$-\nabla\cdot(\varepsilon_0\varepsilon_r\nabla V) = \rho_v$$

where ε_0 is the electrical permittivity of free space, ε_r is the relative electrical permittivity, and ρ_v is the volume charge density, or the quasi-static electric currents equation:

$$-\nabla\cdot((\sigma_e + j\omega\varepsilon_0\varepsilon_r)\nabla V) = 0$$

where σ_e is the electrical conductivity of the material (note that σ is used also for the structural stress vector).

In frequency response analysis the conductivity appears also into the electrostatics equation:

$$-\nabla\cdot\left(\left(\frac{\sigma_e}{j\omega} + \varepsilon_0\varepsilon_r\right)\nabla V\right) = \rho_v$$

and thus you can define and use conductivity of the material independently of the Electrostatics formulation property.

For a *Decoupled, isotropic* material you define ε_r and σ_e as scalars, but for a *Decoupled, anisotropic* material you define them as 3-by-3 matrices.

Electrical Formulations

The default formulation of the equations in the Piezoelectric application modes is such that the resulting equation system with piezoelectric material is symmetric. This allows reduced memory requirements with solvers that utilize symmetry information.

The drawback of this design is that by default the Piezoelectric application modes are not electrically compatible with the Electrostatics application mode found in the AC/DC Module and the MEMS Module, nor is it compatible with the Quasi-Statics -Electric, Electric currents application modes in the AC/DC Module.

The Piezoelectric application modes support an application mode property, Electrostatics formulation, which makes them compatible with the electrostatic or quasi-static application modes so that it is possible to couple them in a model. The Electrostatics formulation property has the following choices:

- Symmetric, Electrostatics: The default implementation creates a symmetric equation system, but the application mode is not compatible with the other application modes.
- Unsymmetric, Electrostatics: This implementation creates an unsymmetric equation system which is compatible with the Electrostatics application modes.
- Unsymmetric, Electric currents: This implementation creates an unsymmetric equation system which is compatible with the Quasistatics - Electric, Electric currents application modes.

At the equation level the difference between these formulation is the following. The default formulation is that the variational electrical energy is written using a positive sign:

$$\delta We = \int (\mathbf{D} \cdot \hat{\mathbf{E}}) d\Omega$$

Here $\mathbf{D}$ is the electric displacement vector, and $\hat{\mathbf{E}}$ is the test function for the Electric field. Ω is the integration domain.

On the other hand, the formulation compatible with the Electrostatics application mode uses variational electrical energy with the negative sign:

$$\delta We = -\int (\mathbf{D} \cdot \hat{\mathbf{E}}) d\Omega$$

Finally, the electric currents formulation uses the following variational electrical energy:

$$\delta We = \int (\mathbf{J} \cdot \nabla \hat{V}) d\Omega$$

where $\mathbf{J}$ is the electric current density vector, and $\nabla \hat{V}$ is the test function for the potential gradient.

The use of the Unsymmetric, electric currents formulation sets certain limitations: you cannot model any charges, and any boundary conditions that use charges or electric displacement are written in terms of electric current. Also, this formulation only appears in the frequency response analysis.

The Piezoelectric Application Modes

This section describes the interface for defining a model using the piezoelectric application modes:

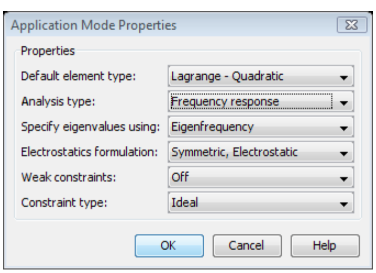
- Piezo Solid (in 3D)
- Piezo Plane Stress (in 2D)
- Piezo Plane Strain (in 2D)
- Piezo Axial Symmetry (in axisymmetric 2D)

It consists of the following sections:

- "Application Mode Properties" (the next section)
- "Scalar Variables" on page 327
- "Material Properties" on page 328
- "Electric Boundary Conditions" on page 338
- "Constraints" on page 342
- "Loads and Charges" on page 344
- "Structural Damping" on page 346

Application Mode Properties

To set or examine material properties, go to the **Physics** menu and open the **Application Mode Properties** dialog box.



Here you control various global settings for the model, which include:

- **Default element type**: A list of elements, where the selection becomes the default on all new subdomains. The default is to use second-order Lagrange elements.

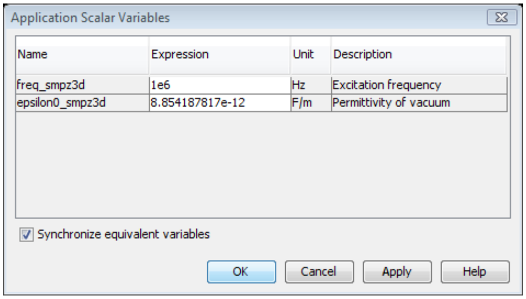
- **Analysis type**: A list of analyses to perform. It affects both the equations and which solver to use with the **Auto select solver** option in the **Solver Parameters** dialog box. The default is static analysis. You can also select transient, eigenfrequency, damped eigenfrequency, and frequency response analysis types.
- **Specify eigenvalues using**: A list controlling whether the application mode works with eigenvalues or eigenfrequencies.
- **Electrostatics formulation**: Select the electrical formulation to use:
  - **Symmetric, Electrostatic**: the default setting.
  - **Unsymmetric, Electrostatic**: for compatibility with the Electrostatics application mode.
  - **Unsymmetric, Electric currents**: for compatibility with the application modes for electric currents in the AC/DC Module (Electric Currents in 3D, In-Plane Electric Currents in 2D, and Meridional Electric Currents in 2D axial symmetry). Available for frequency response analysis.
- **Weak constraints**: Controls whether or not weak constraints are active Use weak constraints for accurate reaction-force computation. When weak constraints are enabled, all constraints are weak by default, but it is possible to change this setting for individual domains.
- **Constraint type**: Constraints can be ideal or nonideal (see “Ideal vs. Non-Ideal Constraints” on page 301 in the *COMSOL Multiphysics Modeling Guide*).

Scalar Variables

The piezoelectric application modes have the following scalar variables:.

PROPERTY	VARIABLE	DEFAULT	SI UNIT	DESCRIPTION
ε_0	`epsilon0`	`8.854187817e-12`	F/m	Permittivity of vacuum
f	`freq`	`1e6`	Hz	Excitation frequency
$j\omega$	`jomega`	`-lambda`	rad/s	Complex angular frequency

You control the scalar variables by going to the **Physics** menu and opening the **Application Scalar Variables** dialog box.



The excitation frequency (the frequency of the harmonic forces, potential, and displacement) is available only for frequency response analysis. The equations and documentation describing frequency response use the angular excitation frequency, $\omega = 2\pi f$, which is available as the variable `omega`. The complex angular frequency is available for eigenfrequency analysis and damped eigenfrequency analysis.

When you select **Frequency response** as the analysis type, the default solver is the parametric solver. This default makes it easy to perform a frequency sweep over several excitation frequencies in one analysis. In this case enter `freq` as the **Parameter name** on the **General** page in the **Solver Parameters** dialog box. The values you enter in the **Parameter values** edit field override the excitation frequency you might have entered in the **Application Scalar Variables** dialog box.

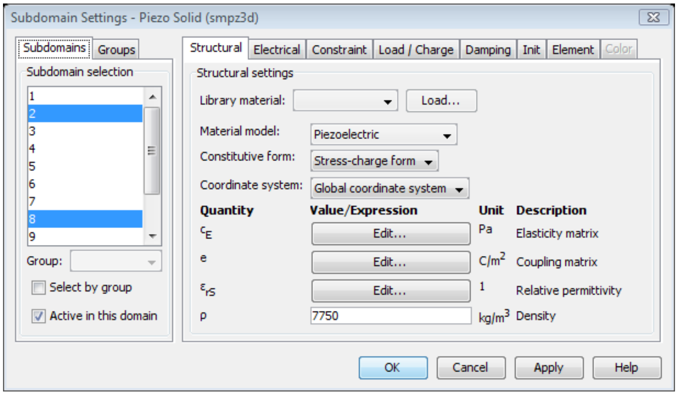
Material Properties

The **Subdomain Settings** window has two pages where you define the material properties: the **Structural** page and the **Electrical** page. On top of both pages you find the **Library material** list and the **Load** button for importing and selecting data from the material libraries and the **Material model** list for selecting the material model for each domain. These settings are shared between the pages, and if you change the **Structural** page, the settings change also on the **Electrical** page. Note that loading a material from a material library does not change the material model, so you need to change it manually in the **Material model** list to match the type of material.

Everything else you see and define on the pages depends on the material model you select. Setting for different material model are described in the following chapters.

SUBDOMAIN SETTINGS FOR PIEZOELECTRIC MATERIAL

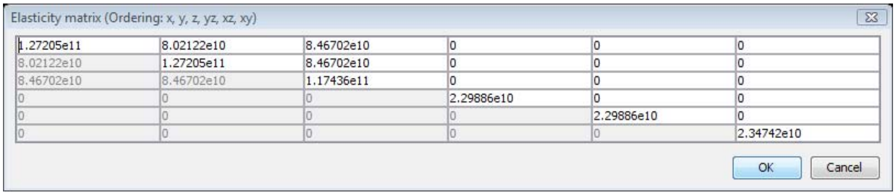
The piezoelectric material is a complete structural-electrical material, and thus you define all piezoelectric material properties on the **Structural** page.



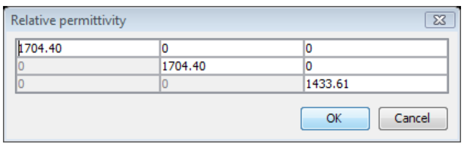
The **Structural** page has two lists in 3D, three lists in 2D, and three lists in axial symmetry:

- **Constitutive form**: Select the constitutive form from those in the following list. Depending on the selection, different material properties are shown in the dialog box.
  - **Stress-charge form**: Define the constitutive relation of the material on the stress-charge form through the c_E, e, and ε_{rS} matrices. The previous figure shows

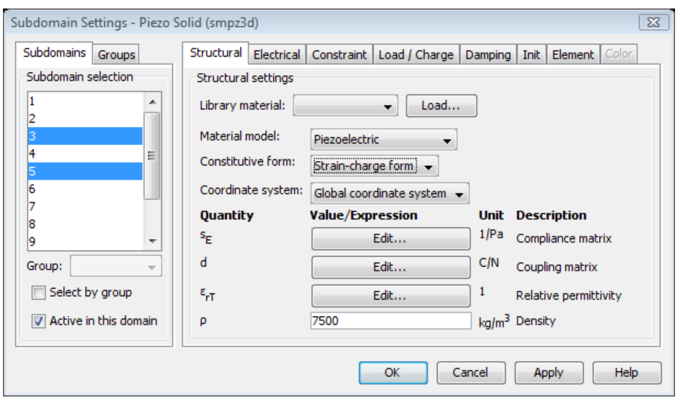
the **Material** page for stress-charge, while the following figure shows the **Elasticity matrix** dialog box for entering the c_E matrix.



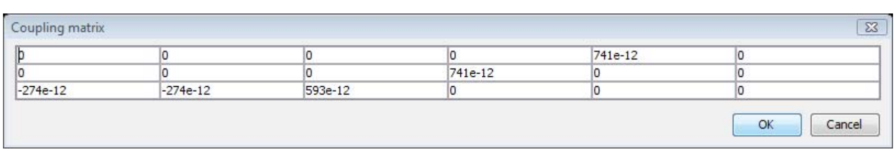
The figure below shows the **Relative permittivity** dialog box for entering the ε_{rS} matrix components.



- **Strain-charge form**: You define the constitutive relation of the material on the strain-charge form through the s_E, d, and ε_{rT} matrices (see page 322 for details). The following figure shows the **Material** page for strain-charge.



The next graphic shows the **Coupling matrix, strain-charge form** dialog box for entering the d matrix components.



- **Material orientation** (2D and axisymmetry only): Here you select how the 3D material properties are oriented relative the 2D/axial symmetric analysis plane. There are six options: *xy*, *yz*, *zx*, *yx*, *zy*, and the default *xz*-plane. The plane represents how the 3D material is oriented relative the 2D/axial symmetric analysis plane: The first letter indicates which 3D direction coincides with the x direction in 2D or the r direction for axisymmetry; the second letter indicates which 3D direction coincides with the y direction in 2D or the z direction for axisymmetry. The material coordinates names are fixed and do not depend of the names of the space coordinates (independent variables), which have different defaults in 2D and axial symmetry.

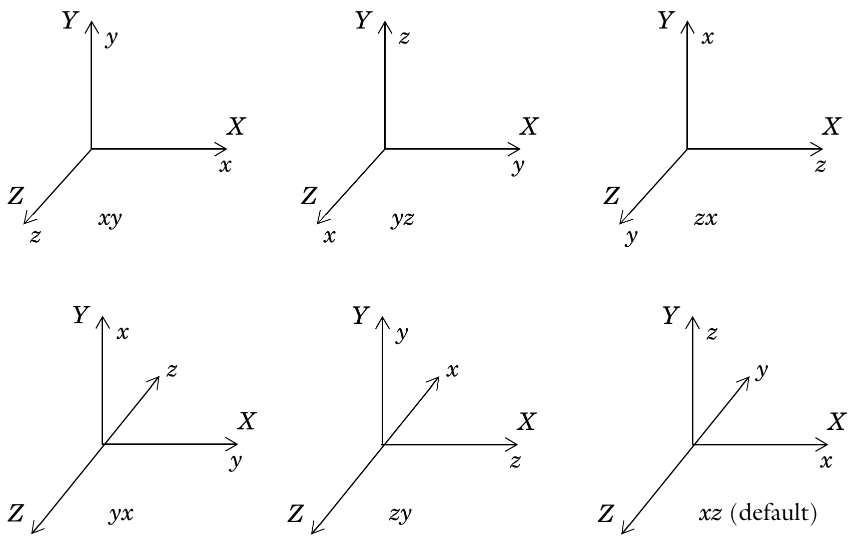


Figure 12-1:Orientation of 3D material xyz relative the 2D analysis coordinate system XYZ.

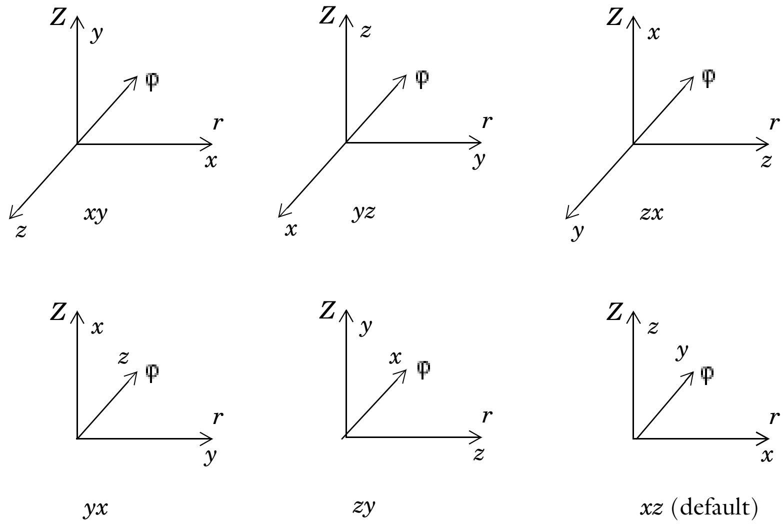


Figure 12-2: Orientation of 3D material xyz relative the axisymmetric analysis coordinate system $r\varphi Z$.

- **Coordinate system**: Select the coordinate system where the material properties are defined. This choice is useful if you want to define the material in a coordinate system other than the global system, or if you need results in a local coordinate system for postprocessing. The **Coordinate system** list contains only the global coordinate system unless you have made available a user-defined coordinate system. You find the **Coordinate System Settings** dialog box on the **Options** menu. Read more about creating a coordinate system and how to use it in "Coordinate Systems" on page 144.

The following table shows the material properties for the union of all constitutive forms and all piezoelectric application modes.

PARAMETER	VARIABLE	DESCRIPTION	CONSTITUTIVE FORM
c_E	`cElk`	Elasticity matrix	Stress-charge
s_E	`sElk`	Compliance matrix	Strain-charge
e	`eik`	Coupling matrix, stress-charge form	Stress-charge
d		Coupling matrix, strain-charge form	Strain-charge

PARAMETER	VARIABLE	DESCRIPTION	CONSTITUTIVE FORM
ε_{rS}		Relative permittivity matrix, stress-charge form	Stress-charge
ε_{rT}		Relative permittivity matrix, strain-charge form	Strain-charge
ρ	`rho`	Density	All
th	`thickness`	Thickness of the geometry (2D only)	All

Elasticity matrix defines the stress-strain relation matrix c_E

$$\sigma = c_E\varepsilon$$

where σ is the stress, and ε is the strain.

Coupling matrix defines the piezo coupling matrix e used in the stress-charge form of the constitutive equation

$$\sigma = c_E\varepsilon - e^T\mathbf{E}$$

where σ is the stress, ε is the strain, and $\mathbf{E}$ is the electric field.

Compliance matrix defines the strain-stress relation matrix s_E

$$\varepsilon = s_E\sigma$$

where σ is the stress, and ε is the strain.

Coupling matrix defines the piezo coupling matrix d used in the strain-charge form of the constitutive equation

$$\varepsilon = s_E\sigma + d^T\mathbf{E}$$

where σ is the stress, ε is the strain, and $\mathbf{E}$ is the electric field.

Relative permittivity the relative permittivity, ε_{rS} and ε_{rT}, appears in the constitutive relation on stress-charge and strain-charge forms, respectively.

$$\mathbf{D} = e\varepsilon + \varepsilon_0\varepsilon_{rS}\mathbf{E}$$

$$\mathbf{D} = d\sigma + \varepsilon_0\varepsilon_{rT}\mathbf{E}$$

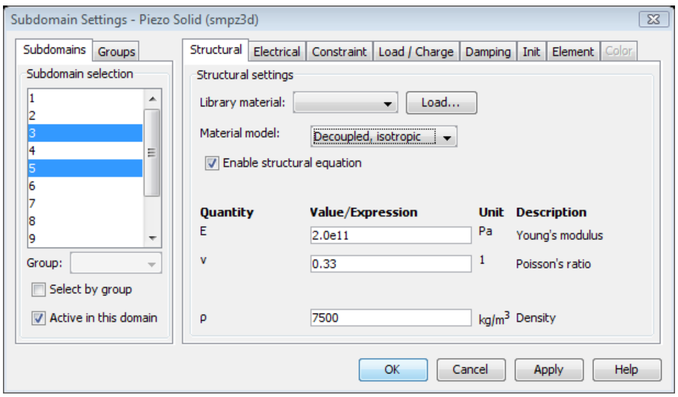
Density this material property, ρ, specifies the material's density.

Thickness this material property, thickness, specifies the material's thickness and appears in 2D only.

SUBDOMAIN SETTINGS FOR DECOUPLED, ISOTROPIC MATERIAL

With this material model you specify material properties on the **Structural** page and the **Electrical** page.

You define the structural material properties on the **Structural** page:



On the first row after the **Material Model** list you find the **Enable structural equation** check box. Use this check box to activate the structural equation or inactivate it to model only electrical problems. By default the **Enable structural equation** check box is selected. If this setting is selected you can define the following structural material properties:

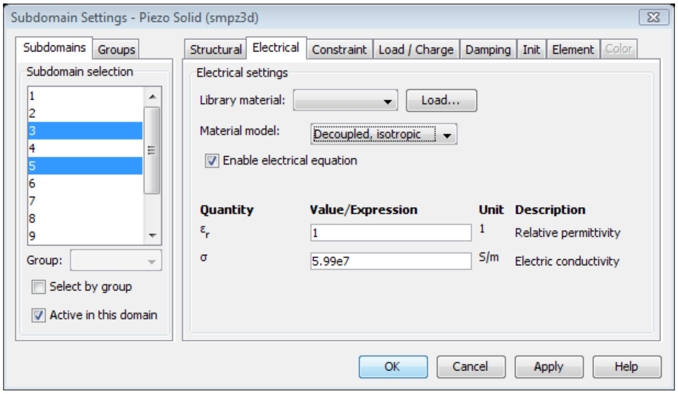
Young's modulus This material property, E, is the modulus of elasticity of the material. It is used to form the elasticity matrix D for the stress strain relationship as described in the chapter "Material Models" on page 322.

Poisson's ratio This material property, ν, defines the contraction of the structure in the perpendicular direction. It is used to form the elasticity matrix D for the stress strain relationship as described in the chapter "Material Models" on page 322.

Density this material property, ρ, specifies the material's density.

Thickness this material property, thickness, specifies the material's thickness and appears in 2D only.

You define the electrical material properties on the **Electrical** page:



On the first row after the **Material Model** list you find the **Enable electrical equation** check box. Use this check box to activate the electrical equation or inactivate it to model only structural problems. If you select it and clear the **Enable structural equation** check box, only the electrical equation is active. By default the **Enable electrical equation** check box is selected. If this setting is selected you can define the following electrical material properties:

Relative permittivity This material property, ε_r, defines the isotropic relative electrical permittivity of the material.

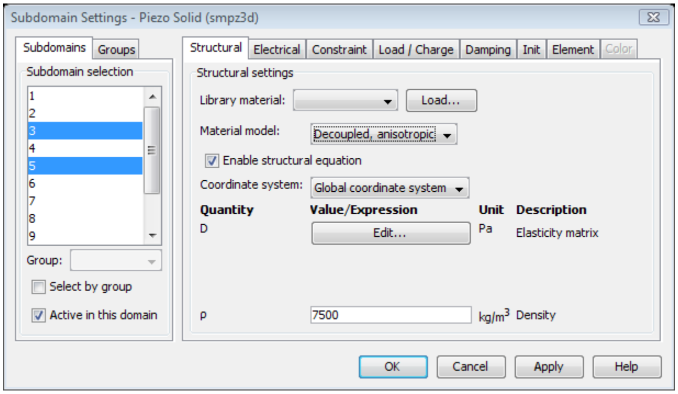
Electric conductivity This material property, σ, defines the isotropic electrical conductivity of the material. This setting only appears for frequency response analysis.

Thickness this material property, thickness, specifies the material's thickness and appears in 2D only.

SUBDOMAIN SETTINGS FOR DECOUPLED, ANISOTROPIC MATERIAL

With this material model you specify material properties on the **Structural** page and the **Electrical** page.

You define the structural material properties on the **Structural** page:

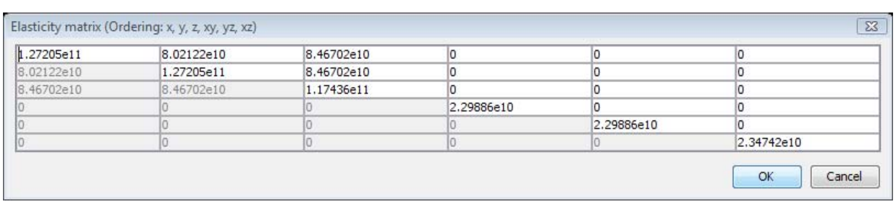


On the first row after the **Material Model** list you find the **Enable structural equation** check box. Use this check box to activate the structural equation or inactivate it to model only electrical problems. By default, **Enable structural equation** is selected. If this setting is selected you can define the following structural material properties:

Material orientation (2D and axisymmetry only): Here you select how the 3D material properties are oriented relative the 2D/axial symmetric analysis plane. There are six options: *xy*, *yz*, *zx*, *yx*, *zy*, and the default *xz*. This setting works the way same as for the piezoelectric material (See description on page 330).

Coordinate system Select the coordinate system where the material properties are defined. This setting works the way same as for the piezoelectric material (See description on page 332).

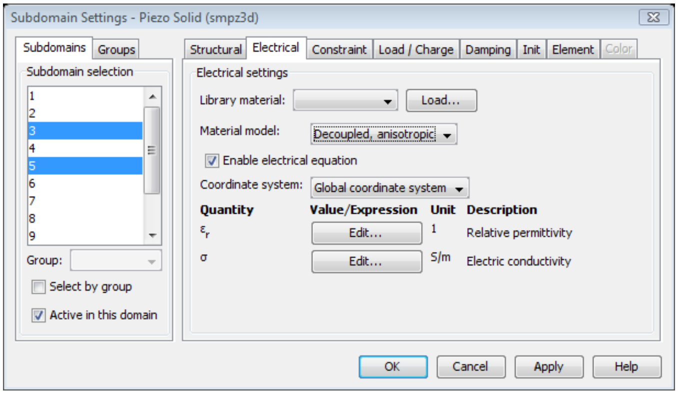
Elasticity matrix This material property, D, defines the elasticity matrix of the anisotropic material (See "Material Models" on page 322.). You define D as a symmetric 6-by-6 matrix:



Density this material property, ρ, specifies the material's density.

Thickness this material property, thickness, specifies the material's thickness and appears in 2D only.

You define the electrical material properties on the **Electrical** page:

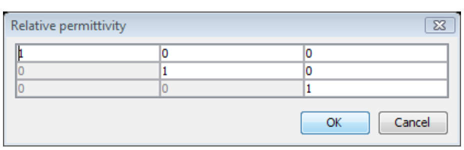


On the first row after the **Material Model** list you find the **Enable electrical equation** check box. Use this check box to activate the electrical equation or inactivate it to model only structural problems. By default **Enable electrical equation** is selected. If this setting is selected you can define the following electrical material properties:

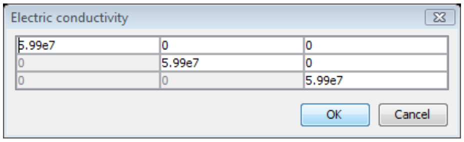
Material orientation (2D and axisymmetry only) This is the same setting as the **Material orientation** in the **Structural** page.

Coordinate system This is the same setting as the **Coordinate system** on the **Structural** page.

Relative permittivity This material property, ε_r, defines the anisotropic relative electrical permittivity of the material. You define ε_r using a symmetric 3-by-3 matrix:



Electric conductivity This material property, σ, defines the anisotropic electrical conductivity of the material. This setting only appears for frequency response analysis. You define σ using a symmetric 3-by-3 matrix:



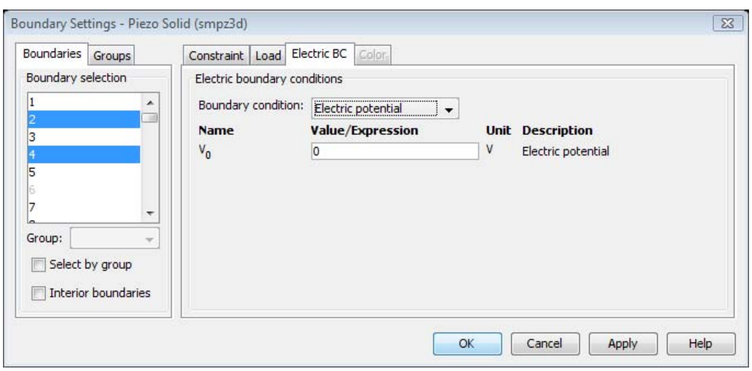
Thickness this material property, thickness, specifies the material's thickness and appears in 2D only.

PIEZOELECTRIC MATERIALS PROPERTIES LIBRARY

A library of about 25 common piezoelectric materials is available through the **Materials/Coefficients Library** dialog box. "Piezoelectric Material Properties Library" on page 110

Electric Boundary Conditions

The electric boundary conditions in the piezoelectric application modes depend on the setting of the **Electrostatics formulation** property in the **Application Mode Properties** dialog box. You specify the electric boundary conditions on the **Electric BC** page in the **Boundary Settings** dialog box.



The **Electric BC** page also has a **Boundary condition** list where you select the type of electric boundary condition; the software enables different edit fields depending on the selected type.

BOUNDARY CONDITIONS FOR ELECTROSTATICS

For the Unsymmetric, Electrostatic and Symmetric, Electrostatic formulations, the boundary conditions include:

Electric Displacement

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

This boundary condition specifies the normal component of the electric displacement at a boundary. Enter the components of the electric displacement $\mathbf{D}_0$.

Surface Charge

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

This boundary condition specifies the surface charge density ρ_s at an exterior boundary (left equation) or at the interior boundary between two media with electric displacement $\mathbf{D}_1$ and $\mathbf{D}_2$, respectively.

Zero Charge/Symmetry

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

This boundary condition specifies that the normal component of the electric displacement is zero. The Zero charge/Symmetry boundary condition is also useful at symmetry boundaries where the potential is symmetric with respect to the boundary.

Electric Potential

$$V = V_0$$

This boundary condition specifies the voltage V_0 at the boundary. Because the application mode computes the electric potential, you must define its value at some boundary in the geometry to be fully determined.

Ground

$$V = 0$$

This boundary condition is a special case of the previous one specifying zero potential. The Ground boundary condition is also be useful at symmetry boundaries, where the potential is antisymmetric with respect to the boundary.

Continuity

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

This boundary condition specifies that the normal component of the electric displacement is continuous across an interior boundary or across a boundary between a piezoelectric and an electrostatic domain if you use the Unsymmetric, Electrostatic formulation. Using the Symmetric, Electrostatic formulation the Continuity condition is only available for interior boundaries, where it is the default.

Floating Potential

This condition the potential on the boundary to a spatially constant value such that the total charge on the boundary equals the user defined total charge Q_0:

$$\int_{\partial\Omega} \rho_s = Q_0$$

You also define the group index, which defines how the boundaries are grouped in to a set of electrodes.

Axial Symmetry

$$E_r = 0$$

$$\frac{\partial E_z}{\partial r} = 0$$

This boundary condition is the natural Neumann boundary condition, which you use on the z-axis ($r = 0$) to maintain the symmetry conditions. The Axial Symmetry boundary condition is available only in the Piezo Axial Symmetry application mode.

BOUNDARY CONDITIONS FOR ELECTRIC CURRENTS

For the Unsymmetric, Electric currents formulations, the boundary conditions include:

Ground

$$V = 0$$

This boundary condition is a special case of the previous one specifying zero potential. The Ground boundary condition is also be useful at symmetry boundaries, where the potential is antisymmetric with respect to the boundary.

Electric Potential

$$V = V_0$$

This boundary condition specifies the voltage V_0 at the boundary. Because the application mode computes the electric potential, you must define its value at some boundary in the geometry to be fully determined.

Current Flow

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

This boundary condition specifies the current flow. Enter the components of the current density $\mathbf{J}_0$.

Inward Current Flow

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

This boundary condition specifies the normal current density J_n at an exterior boundary.

Electric Insulation

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

This boundary condition specifies that the normal component of the electric current is zero; that is, the boundary is electrically insulated.

Current Source

The current source boundary condition

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

is applicable to interior boundaries that represent either a source or a sink of current.

Continuity

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

This boundary condition specifies that the normal component of the electric current is continuous across the interior boundary (where it is the default setting) or across a boundary between a piezoelectric and an domain with electric currents.

Floating Potential

This condition the potential on the boundary to a spatially constant value such that the total current through the boundary equals the user defined total current I_0:

$$\int_{\partial\Omega} -\mathbf{n} \cdot \mathbf{J} = I_0$$

You also define the group index, which defines how the boundaries are grouped in to a set of electrodes.

Axial Symmetry

This boundary condition is the natural Neumann boundary condition, which you use on the z-axis ($r = 0$) to maintain the symmetry conditions. The Axial Symmetry boundary condition is available only in the Piezo Axial Symmetry application mode.

CONVERSION OF ELECTRIC BOUNDARY CONDITIONS

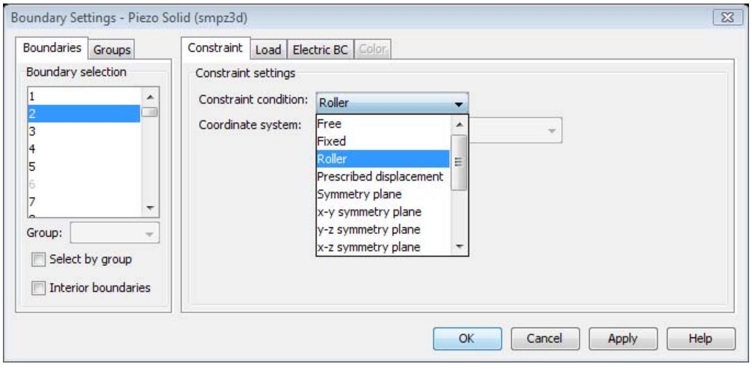
Some boundary conditions are applicable only for the formulations for electrostatics, whereas others apply only to the formulation for electric currents. Table 12-1 contains the boundary conditions that the software converts when changing from one formulation to the other:

TABLE 12-1: BOUNDARY CONDITION CONVERSIONS

BOUNDARY CONDITION FOR ELECTROSTATICS	BOUNDARY CONDITION FOR ELECTRIC CURRENTS
Electric displacement	Current flow
Zero charge/Symmetry	Electric insulation
Surface charge (exterior boundaries)	Inward current flow
Surface charge (interior boundaries)	Current source

Constraints

A constraint specifies the displacement or potential of certain parts of a structure. You can define constraints for the displacements on all domain levels including points, edges, faces/boundaries, and subdomains (in 3D), and points, boundaries, and subdomains (in 2D). In addition, you can define constraints for the potential on points and edges in 3D, and for points in 2D. To control them, go to the **Constraint** page in the **Subdomain/Boundary/Edge/Point Settings** dialog boxes, and set constraints on boundaries from the **Electric BC** page. The following figure shows the **Boundary Settings** dialog box for the Piezo Solid application mode, but the page has the same appearance in all piezoelectric application modes.

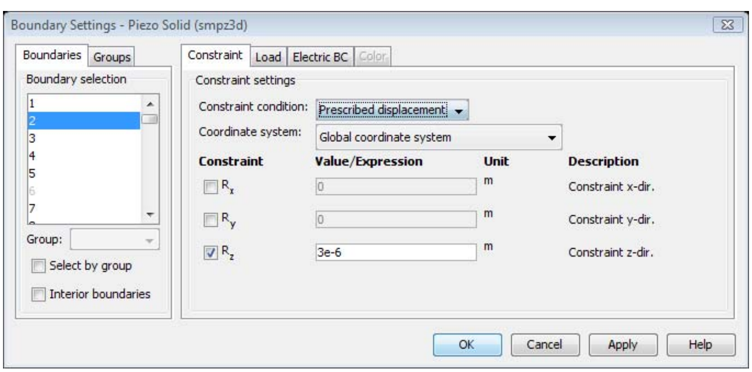


Use the **Constraint condition** list in this dialog box to select the type of constraint that you want to define. See “Constraints” on page 79 for details.

The **Coordinate system** list lets you control in which coordinate system you want the constraint defined. Available options are:

- Global coordinate system
- Tangent and normal coordinate system, available only on boundaries
- User-defined coordinate systems, if any local coordinate systems are defined. (Read more about creating a coordinate system in the section “Coordinate Systems” on page 144.)

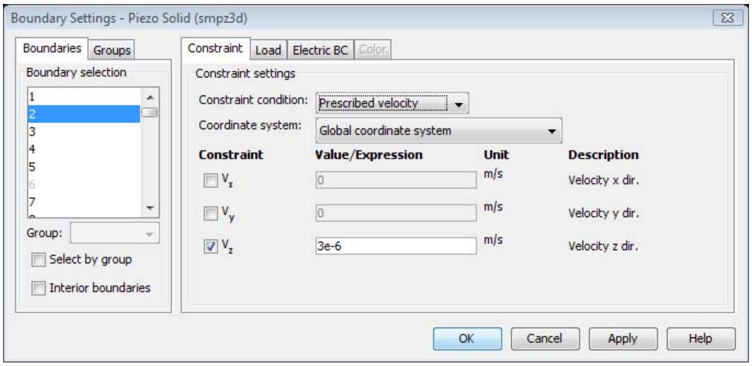
When you select **Prescribed displacement** a number of new options appears in the dialog box and the **Constraint** page takes on this appearance:



The Constraint page showing the prescribed displacement options.

The check boxes adjacent to the $\mathbf{R}_x$, $\mathbf{R}_y$, and $\mathbf{R}_z$ edit fields activate the constraint, whereupon you enter the value/expression of the displacement (the default value is `0`).

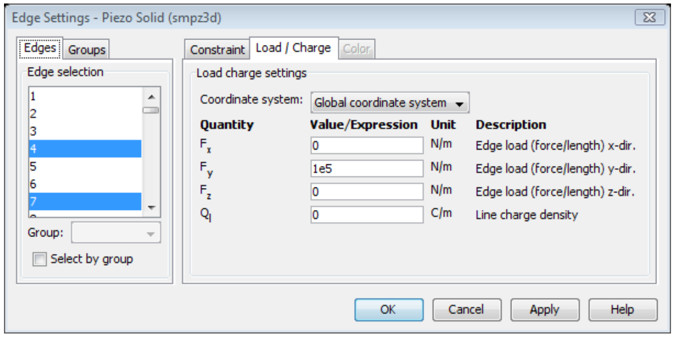
In a frequency response analysis you have the possibility to specify not only a harmonic displacement but also a harmonic velocity or acceleration. You specify the **Prescribed velocity** and **Prescribed acceleration** in the same way as **Prescribed displacement**.



Constraint page showing the prescribed velocity settings.

Loads and Charges

Load is a general name for forces applied to a structure. You can specify loads on all domain types. To do so, click the **Load** tab in the **Boundary Settings** dialog boxes or the **Load/Charge** tab in the **Subdomain Settings**, **Edge Settings**, and **Point Settings** dialog boxes, where you can also specify a charge density. The formulation for electric currents does not include charges, so in that case, the name of the tab is **Load** also in the **Subdomain Settings**, **Edge Settings**, and **Point Settings** dialog boxes. The following image shows the **Edge Settings** dialog box for the Piezo Solid application mode, but the tab looks similar on all domain levels in all piezoelectric application modes.



SPECIFYING LOADS

For plane stress and plane strain, option buttons allow you to specify the load in different ways using the thickness. The following table summarizes the options for defining loads on different domains in different application modes; the SI unit appears in parenthesis.

APPLICATION MODE	POINT	EDGE	BOUNDARY	SUBDOMAIN
Plane Stress, Plane Strain	force (N)		force/area (N/m^2) or force/length (N/m)	force/volume (N/m^3) or force/area (N/m^2)
Axial symmetry	total force along the circumferential (N)		force/area (N/m^2)	force/volume (N/m^3)
Solid	force (N)	force/length (N/m)	force/area (N/m^2)	force/volume (N/m^3)

With the **Coordinate system** list you control in which coordinate system the load is defined. Available options are:

- Global coordinate system
- Tangent and normal coordinate system, only available on boundaries
- User-defined coordinate systems, if there are any local coordinate systems defined. Read more about creation of coordinate system in the coordinate system section.

SPECIFYING CHARGES

You can specify a charge on the **Edge/Point** level when you use a formulation for electrostatics. For plane stress and plane strain, option buttons allow you to specify the charge in different ways using the thickness. The following table summarizes the options for defining charge on different domains in different application modes; the SI units appears in parenthesis.

APPLICATION MODE	POINT	EDGE	SUBDOMAIN
Plane Stress, Plane Strain	charge (C)		charge/volume (C/m^3) or charge/area (C/m^2)
Axial symmetry	total charge along the circumferential (C)		charge density (C/m^3)
Solid	force (C)	charge/length (C/m)	charge density (C/m^3)

To specify charge density on boundaries, click the **Electric BC** tab.

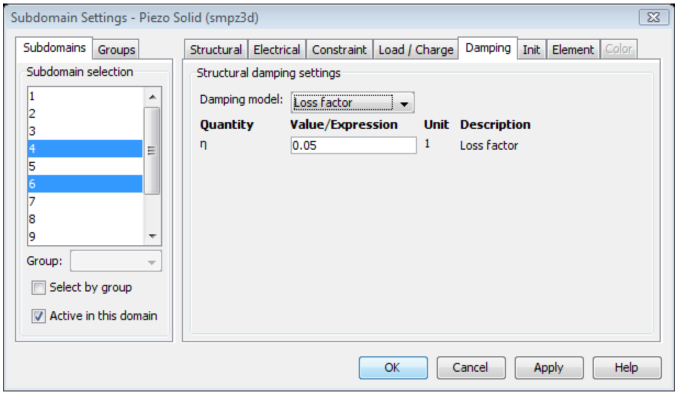
Structural Damping

For time-dependent analysis, you can specify viscous damping (structural damping) using Rayleigh damping, where the damping matrix is specified to be proportional to the mass and stiffness matrix:

$$C = \alpha_{dM}M + \beta_{dK}K$$

For frequency response analysis you can specify viscous damping using either Rayleigh damping, loss factor damping, or equivalent viscous damping.

To specify structural damping parameters, go to the **Damping** page in the **Subdomain Settings** dialog box, and choose the type of damping model from the **Damping model** list. The layout of the dialog box changes for each damping model.



The Damping page when loss factor damping is selected.

Note: Loss factor damping and equivalent viscous damping are valid only for frequency response analysis. If you choose a transient analysis and either of these damping types, COMSOL Multiphysics solves the model with no damping.

Table 12-2 and the following text describe the parameters that define damping:

TABLE 12-2: PARAMETERS FOR DAMPING MODELS

PARAMETER	VARIABLE	DESCRIPTION	DAMPING MODEL
α_{dM}	`alphadM`	Mass-damping parameter	Rayleigh
β_{dK}	`betadK`	Stiffness-damping parameter	Rayleigh
η	`eta`	Loss factor	Loss factor, Equivalent viscous

Mass damping parameter Defines the Rayleigh damping model's mass damping, α_{dM}.

Stiffness damping parameter Defines the Rayleigh damping model's stiffness damping, β_{dK}.

Loss factor Defines the loss factor η for the loss factor damping and equivalent viscous damping models.

The Piezo Solid Application Mode

Use the Piezo Solid application mode for analysis of 3D structures that exhibit piezoelectric effects.

VARIABLES AND SPACE DIMENSIONS

The degrees of freedom (dependent variables) are the global displacements u, v, and w in the global x, y, and z directions, and the electric potential, V.

PDE FORMULATION

The implementation of this application mode uses the principle of virtual work, described in general terms in the section "Implementation" on page 181.

APPLICATION MODE VARIABLES

For information about available application mode variables, see "Piezoelectric Application Modes" on page 58 in the *Structural Mechanics Module Reference Guide*.

The Piezo Plane Stress Application Mode

Use the Piezo Plane Stress application mode to analyze thin in-plane loaded plates that exhibit piezoelectric effects.

VARIABLES AND SPACE DIMENSIONS

The degrees of freedom (dependent variables) are the global displacements u and v in the global x and y directions, and the electric potential V.

PDE FORMULATION

The implementation of this application mode uses the principle of virtual work, which this manual describes in general terms in the section "Implementation" on page 181.

Application Mode Parameters

For details about the application mode parameters that define the loads, charges, material properties, constraints, and electric boundary conditions, see the sections earlier in this chapter.

APPLICATION MODE VARIABLES

For information about available application mode variables, see "Piezoelectric Application Modes" on page 58 in the *Structural Mechanics Module Reference Guide*.

The Piezo Plane Strain Application Mode

Use the Piezo Plane Strain application mode to compute the global displacements (u, v) in the x and y directions and the electric potential for a piezoelectric structure in a state of plane strain. The plane strain condition assumes that the ε_z, ε_{yz}, and ε_{xz} components of the strain tensor are zero.

VARIABLES AND SPACE DIMENSIONS

The degrees of freedom (dependent variables) are the global displacements u and v in the global x and y directions, and the electric potential V.

PDE FORMULATION

The implementation of this application mode uses the principle of virtual work, described in general terms in the section “Implementation” on page 181.*Application Mode Parameters*

For details about the application mode parameters that define the loads, charges, material properties, constraints, and electric boundary conditions, see the sections earlier in this chapter.

APPLICATION MODE VARIABLES

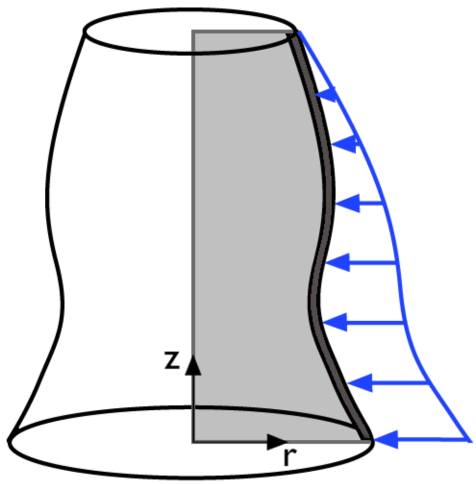
For information about available application mode variables, see “Piezoelectric Application Modes” on page 58 in the *Structural Mechanics Module Reference Guide*.

The Piezo Axial Symmetry Application Mode

Use the Piezo Axial Symmetry application mode to analyze axisymmetric models of materials showing piezoelectric effects.

This application mode uses cylindrical the coordinates r, φ (`phi`), and z. It solves the equations for the global displacement (u, w) in the r and z directions. It assumes that the displacement v in the φ direction together with the $\tau_{r\varphi}$, $\tau_{\varphi z}$, $\gamma_{r\varphi}$, and $\gamma_{\varphi z}$ components of the stresses and strains are zero. Loads are independent of φ, and it allows loads only in the r and z directions.

You can consider the domain where the software solves the equations as the intersection between the original axially symmetric 3D solid and the half plane $\varphi = 0$, $r \geq 0$. Therefore it is necessary to draw the geometry only in the half plane $r \geq 0$. The software recovers the original 3D solid by rotating the 2D geometry about the z-axis as seen in the following figure:



The strain-displacement relations for the axial symmetry case for small displacements are:

$$\varepsilon_r = \frac{\partial u}{\partial r} \qquad \varepsilon_\varphi = \frac{u}{r} \qquad \varepsilon_z = \frac{\partial w}{\partial z} \qquad \gamma_{rz} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial r}$$

To avoid division by r (which causes problems on the axis, where $r = 0$), the program automatically transforms the equations by multiplying by r. When using the principle of virtual work, you normally do not think of this multiplication as a transformation but merely as an integration around the circumference. Integrating over the volume, you must multiply the integrand by $2\pi r$. The application mode introduces and solves for a new dependent variable

$$\texttt{uor} = \frac{u}{r}$$

instead of the true radial displacement, u.

Note: $r = 0$ is the symmetry axis. x -> r and y -> z in the Piezo Axial Symmetry, application mode.

VARIABLES AND SPACE DIMENSIONS

The degrees of freedom (dependent variables) are uor the radial displacement divided by r and w the global displacement in the z direction and the electric potential V.

PDE FORMULATION

The implementation of this application mode uses the principle of virtual work, described in general terms in the section "Implementation" on page 181.

Application Mode Parameters

For details about the application mode parameters that define the loads, charges, material properties, constraints, and electric boundary conditions, see the sections earlier in this chapter.

APPLICATION MODE VARIABLES

For information about available application mode variables, see "Piezoelectric Application Modes" on page 58 in the *Structural Mechanics Module Reference Guide*.

13

Predefined Multiphysics Couplings

The Structural Mechanics Module contains predefined multiphysics couplings to facilitate easy set up of models with the most commonly occurring couplings. These predefined multiphysics couplings automatically add the necessary application modes with appropriate settings to your model and define the applicable couplings for the interaction between the different types of physics.

Thermal-Structure Interaction

The Thermal-Structure Interaction predefined multiphysics coupling combines a continuum application mode from the Structural Mechanics Module with a heat transfer application mode from the Heat Transfer Module or COMSOL Multiphysics. The coupling appears on the subdomain level, where the temperature from the heat transfer application mode acts as a thermal load for the structural mechanics application mode.

Theory Background

Read about constitutive equations including thermal expansion in the section dealing with the theory background for the continuum application modes, on page 164 of this manual.

Application Mode Description

A combination of the following two application modes make up the Thermal-Structure Interaction predefined coupling:

- A continuum application mode from the Structural Mechanics Module:
  - Plane Strain or Plane Stress in 2D
  - Axial Symmetry, Stress-Strain in 2D axisymmetry
  - Solid, Stress-Strain in 3D
- The General Heat Transfer application mode from the Heat Transfer Module, if your license includes that module, or the Heat Transfer, Conduction application mode from COMSOL Multiphysics

This section describes settings specific to the Thermal-Structure Interaction predefined multiphysics coupling. Use the following table to locate the sections describing the individual application modes.

APPLICATION MODES	APPLICATION MODE DESCRIPTION
Continuum application modes, Structural Mechanics Module	page 159

APPLICATION MODES	APPLICATION MODE DESCRIPTION
General Heat Transfer, Heat Transfer Module	page 22 in the *Heat Transfer Module User's Guide*
Heat Transfer, Conduction, COMSOL Multiphysics	page 167 in the *COMSOL Multiphysics Modeling Guide*

ANALYSIS TYPE

There are three available analysis types in the Model Navigator: a static analysis type, which uses the stationary solver, and the transient and quasi-static analysis types, which use the time-dependent solver.

SUBDOMAIN SETTINGS

Both application modes of this predefined multiphysics coupling are active on all subdomains of the model. Thermal expansion is enabled for all subdomains in the structural mechanics application mode. In the graphical user interface, you can find this on the **Load** page of the **Subdomain Settings** dialog box for the structural mechanics application mode, where the predefined coupling automatically selects the **Include thermal expansion** check box. On the same page, the expression in the **Temp** edit field is the dependent variable for temperature from the heat transfer application mode, typically `T`.

BOUNDARY SETTINGS

The Thermal-Structure Interaction predefined coupling does not define any coupled constraints or loads on the boundaries. You can set those individually for the structural and thermal analyses.

Example Model

See Chapter 15, "Thermal-Structure Interaction," of the *Structural Mechanics Module Model Library* for models that exemplify thermal-structure interaction.

Fluid-Structure Interaction

The Fluid-Structure Interaction (FSI) predefined multiphysics coupling combines fluid flow with structural mechanics by using a Moving Mesh (ALE) application mode to capture the movement of the fluid domain. The structural mechanics application mode uses the large-deformation option, and the fluid flow application mode enables weak constraints that provide the fluid loads on the structure. The fluid flow application mode is defined on an ALE frame, whereas the structural mechanics application mode for the solid is defined on a reference frame. The FSI couplings appear on the boundaries between the fluid and the solid, and there are also predefined settings for the subdomain properties. These settings are grouped into easily identifiable groups that you assign to the relevant subdomains and boundaries.

Theory Background

The fluid flow is described by the Navier-Stokes equations (Equation 6-1 in the *COMSOL Multiphysics Modeling Guide*), which provide a solution for the velocity field **u**. The total force exerted on the solid boundary by the fluid is the negative of the reaction force on the fluid,

$$\mathbf{f} = -\mathbf{n}\cdot(p\mathbf{I}+\eta(\nabla\mathbf{u}+(\nabla\mathbf{u})^T)) \tag{13-1}$$

where p denotes pressure, η the dynamic viscosity for the fluid, $\mathbf{n}$ the outward normal to the boundary, and $\mathbf{I}$ the identity matrix. Because the Navier-Stokes equations are solved in the spatial (deformed) coordinate system while the structural mechanics application modes are defined in the reference (undeformed) coordinate system, a transformation of the force is necessary. This is done according to

$$\mathbf{F} = \mathbf{f}\cdot\frac{dv}{dV} \tag{13-2}$$

where dv and dV are the mesh element scale factors for the spatial frame and the reference frame, respectively.

The FSI predefined multiphysics coupling computes the reaction force on the fluid by turning on the weak constraints option for the fluid application mode, which adds Lagrange multipliers as additional dependent variables. Due to the derivatives present in the boundary condition for the velocity field, non-ideal weak constraints are used.

You can read about weak constraints in the section "Using Weak Constraints" on page 300 of the *COMSOL Multiphysics Modeling Guide*.

Application Mode Description

A combination of the following three application modes make up the FSI predefined multiphysics coupling:

- A continuum application mode from the Structural Mechanics Module:
  - Plane Strain in 2D
  - Axial Symmetry, Stress-Strain in 2D axisymmetry
  - Solid, Stress-Strain in 3D
- Moving Mesh (ALE) from COMSOL Multiphysics
- The Incompressible Navier-Stokes application mode from the Chemical Engineering Module, if the license includes that module, or from COMSOL Multiphysics

This section describes settings specific to the FSI predefined coupling. Use the following table to locate the sections describing the individual application modes.

APPLICATION MODES	APPLICATION MODE DESCRIPTION
Continuum application modes, Structural Mechanics Module	page 159
Moving Mesh (ALE), COMSOL Multiphysics	page 391 in the *COMSOL Multiphysics Modeling Guide*
Incompressible Navier-Stokes, COMSOL Multiphysics	page 130 in the *COMSOL Multiphysics Modeling Guide*

PROPERTIES

The FSI predefined multiphysics coupling change some of the application mode properties from their default settings according to the following table:

APPLICATION MODE	PROPERTY	SETTING
Continuum application modes	**Large deformation**	**On**
Moving Mesh (ALE)	**Smoothing method**	**Winslow**
	Weak constraint	**Off**
Incompressible Navier-Stokes	**Constraint type**	**Non-ideal**

ANALYSIS TYPE

There are two available analysis types in the Model Navigator, a static analysis that uses the stationary solver and a transient analysis that uses the time-dependent solver.

SUBDOMAIN SETTINGS

From within the **Subdomain Settings** dialog box for each application mode, you can assign a group of settings to each subdomain by selecting it from the **Group** list. The following groups are available:

- **Fluid domain**. This group contains subdomain settings for the fluid domain.
  - In the structural mechanics application mode, this group makes this application mode inactive.
  - In the Incompressible Navier-Stokes application mode, this group uses the default properties for the fluid. Change these properties to match the fluid in your model.
  - In the Moving Mesh (ALE) application mode, this group defines free mesh displacement
- **Solid domain**. This group contains subdomain settings for the solid domain.
  - In the structural mechanics application mode, this group uses the default properties for the solid. Change these properties to match the solid in your model.
  - In the fluid flow application mode, this group makes this application mode inactive.
  - In the Moving Mesh (ALE) application mode, this group defines physics-induced mesh displacement using the displacements from the structural mechanics application mode. Note that in 2D axisymmetry, these displacements are defined as prescribed mesh displacements because the dependent variables in the Axial Symmetry, Stress-Strain application mode differ from the actual displacements, which instead are available as variables. The following table shows the applied settings for the different space dimensions.

SPACE DIMENSION	SELECTION	EDIT FIELD	EXPRESSION
2D	**Physics induced displacement**	**dx**	u
		dy	v
3D	**Physics induced displacement**	**dx**	u
		dy	v
		dz	w

SPACE DIMENSION	SELECTION	EDIT FIELD	EXPRESSION
2D axial symmetry	**Prescribed displacement**	**dr**	`uaxi_smaxi`
		dz	`w`

BOUNDARY SETTINGS

You can apply predefined boundary settings by selecting a group from the **Group** list in the **Boundary Settings** dialog box for the application modes. Each of the following groups is available only in one of the application modes:

- **Fluid load**. This group, found in the structural mechanics application mode, defines the fluid load on the structure using the variable for the total force per area times a factor for the area effect, for example, `T_x_ns*dvol_ale/dvol`. The expression includes a factor for the area effect because the total force variable comes from the deformed mesh, whereas the forces in the structural mechanics application mode must be based on the undeformed area. This factor is the mesh element scale factor for the ALE frame divided by the mesh element scale factor for the reference frame. Also, for axisymmetric models, an additional factor `(R+uaxi)/R` takes the radial displacement into account.
- **Structural velocity**. This group, found in the fluid flow application mode, is only applicable for transient analysis, where the time derivatives of the structural displacements define the fluid's velocity. **Moving leaking wall** is set as **Boundary condition** with components according to the following table.

SPACE DIMENSION	EDIT FIELD	EXPRESSION
2D	u_w	`ut`
	v_w	`vt`
3D	u_w	`ut`
	v_w	`vt`
	w_w	`wt`
2D axial symmetry	u_w	`uaxi_t_smaxi`
	v_w	`wt`

- **Structural displacement**. Use this setting in the Moving Mesh (ALE) application mode at the boundaries of the solid domain. The settings define the mesh displacements as the structural displacements, according to the table below.

SPACE DIMENSION	EDIT FIELD	EXPRESSION
2D	dx	`u`
	dy	`v`
3D	dx	`u`
	dy	`v`
	dz	`w`
2D axial symmetry	dr	`uaxi_smaxi`
	dz	`w`

- **Fixed**. This group, found in the Moving Mesh (ALE) application mode, defines the mesh displacements to be zero. Use this setting at the exterior boundaries of the fluid domain.

In addition to the above predefined settings, you typically define standard boundary conditions such an inflow velocities, slip, and no-slip conditions in the fluid flow application mode and one or several fixed boundaries in the structural mechanics application mode.

Example Model

"Obstacle in Fluid" on page 417 in the *Structural Mechanics Module Model Library* demonstrates a 3D static FSI simulation.

14

Fatigue Analysis

This chapter describes how to perform fatigue analysis using the Structural Mechanics Module together with COMSOL Script or MATLAB.

Background and Introduction to Fatigue Analysis

The term *fatigue* is used for describing the phenomenon where a component fails after repeated loadings and unloadings, even though the magnitude of each individual load is smaller than the ultimate stress of the material. The term was coined in the middle of the nineteenth century, when a number of railroad accidents draw attention to the subject. A vast majority of all structural failures even today are attributed to fatigue, so dimensioning against fatigue is of the utmost importance.

When a fatigue failure occurs, the process can be divided into three stages:

1. During a large number of *load cycles* (repeated loadings and unloadings), damage is accumulated on the micromechanical scale, and after some time a crack of macroscopic size is formed.
2. The macroscopic crack grows for each new load cycle.
3. When the crack has reached a certain size, the remaining material can no longer sustain the peak load, and the component fails.

Usually, the last two stages are considered within the topic of *fracture mechanics,* and the term fatigue applies mainly to Stage 1. Because the largest part of the life of the component is spent before it is possible to observe a macroscopic crack, most designs aim to avoid ever getting such a crack.

Phenomenology and Testing

The underlying reason for fatigue must be sought on the micromechanical scale, on which materials are not homogenous. In an alloy there are grains, whose boundaries cause stress concentrations. In a casting, there might even be pores that are formed during the solidification. Thus, on a local scale, the strains might be much larger than their macroscopic average values, and dislocations within the crystals are activated.

Because the location of these micromechanical irregularities are more or less randomly distributed, there is a large scatter in the number of cycles that a certain type of component can be subjected to, even if the external load is well defined. This scatter makes it necessary to test many specimens when looking for fatigue data. Two examples of these statistical effects are:

- If two sets of bars with different diameters are tested in tension with the same nominal stress, the larger one will appear to have a shorter lifetime. The reason is that within a larger volume of material, the risk of finding a microscopic defect of a certain size is larger.
- If the same type of bar is tested in both tension and bending giving the same peak stress, the one tested in bending will appear to have longer lifetime. During bending only a small volume of the material is subjected to the highest stress.

A pioneer in the field of fatigue was the German engineer August Wöhler who presented a classical work in 1870. His name is used for diagrams showing *stress amplitude* (see Equation 14-2 on page 364) versus number of cycles to fatigue. They are called *Wöhler curves* or *S-N curves.* An example appears in Figure 14-1

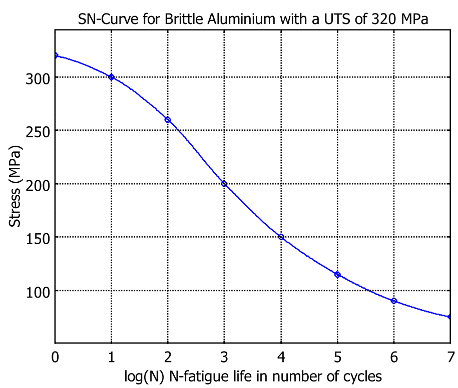


Figure 14-1: Example of an S-N curve.

You usually obtain an S-N curve by testing at different stress levels and recording the number of cycles to failure. Several specimens are tested at each level, so that average and scatter can be computed, giving one point of the curve. Note that because the S-N curve gives the level at which a certain percentage (often 50%) of a population can be expected to fail, that value cannot always be directly used for dimensioning. Knowledge of the scatter in terms of the standard deviation is necessary to transform the given data to another, acceptable, level. For a certain number of cycles, a certain stress level is then connected to a probability of failure. The acceptable probability (and thus stress level) in a design of course differs between a passenger aircraft and a lawn mower.

There are two different regimes of the fatigue phenomena: Low-cycle fatigue (LCF) and high-cycle fatigue (HCF). The border between the two is in no way exact, but usually a cycle count larger than 10^4 is considered as “high cycle.” Characteristic of LCF is that significant plastic strains occur on the macroscopic scale.

For some materials (for example, many steel and titanium alloys) a lowest stress level exists, below which fatigue does not occur irrespective of the number of load cycles. This level is called the *fatigue limit* or *endurance limit*. Many other materials such as aluminum and copper do not appear to have such a limit.

For a material having a fatigue limit, the S-N curve has a horizontal asymptote at large cycle numbers. Fatigue limits are often of the order of half the ultimate tensile strength.

Even for a material without a fatigue limit, such values are sometimes given. They actually represent the value of the S-N curve at a certain large number of cycles, for example, $5 \cdot 10^7$.

It is often possible to represent the central part of the S-N curve (which is in the HCF regime) by a straight line in a log-log diagram. This relation is called the *Basquin equation*, which states that

$$\sigma_a = \sigma'_f (2N_f)^b \tag{14-1}$$

Here N_f is the number of load reversals, so that $2N_f$ is the number of full cycles. σ'_f and b are material parameters.

DEFINITIONS OF FATIGUE QUANTITIES

In addition to the stress amplitude, the mean stress is also important for when fatigue cracks appear. A tensile mean stress decreases the fatigue life, while a compressive stress increases it. If σ_{max} is the maximum stress over the cycle, and σ_{min} is the minimum stress, the following definitions are used:

Stress amplitude:

$$\sigma_a = \frac{\sigma_{max} - \sigma_{min}}{2} \tag{14-2}$$

Stress range

$$\Delta\sigma = \sigma_{max} - \sigma_{min} \tag{14-3}$$

Mean stress:

$$\sigma_m = \frac{\sigma_{\max} + \sigma_{\min}}{2} \tag{14-4}$$

R-value:

$$R = \frac{\sigma_{\min}}{\sigma_{\max}} \tag{14-5}$$

The R-value is the most commonly used parameter for describing the mean stress level.

The most common fatigue test is the one where the loading is fully reversed, that is having a zero mean stress ($R = -1$). The second fundamental test is the pulsating test, where the load varies between zero and a maximum value ($R = 0$). For cases with nonzero mean stresses, note that the S-N curve can be defined in terms of either the stress amplitude or the maximum stress. The functions used in the Structural Mechanics Module is specified through the stress amplitude and R-value.

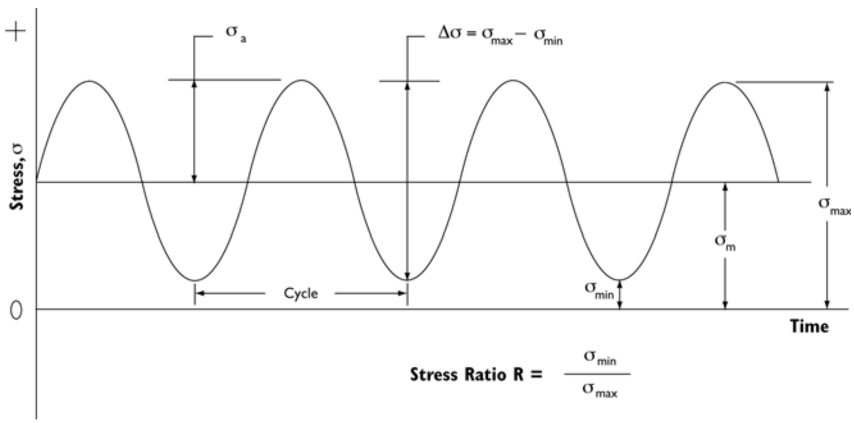


Figure 14-2: Example of cyclic loading.

MEAN STRESS EFFECTS

The mean stress effects can be represented in a *Haigh diagram*, a plot of the stress amplitude versus the mean stress for different number of load cycles (see Figure 14-3). Unfortunately, it is rare that enough data are available, so in practice rather crude simplifications are made. The most common mean stress corrections are the Goodman and the Gerber corrections. The Goodman correction approximates the curve in the Haigh diagram by a straight line, and the Gerber correction approximates it by a parabola. The simplified diagram appears in Figure 14-4. If data is available for both

$R = -1$ and $R = 0$, it is possible to use the bilinear approximation that Figure 14-4 also includes.

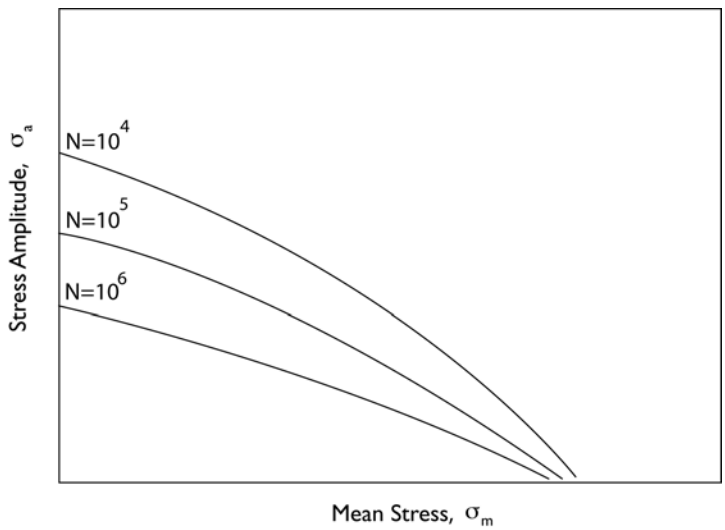


Figure 14-3: An example of a Haigh diagram.

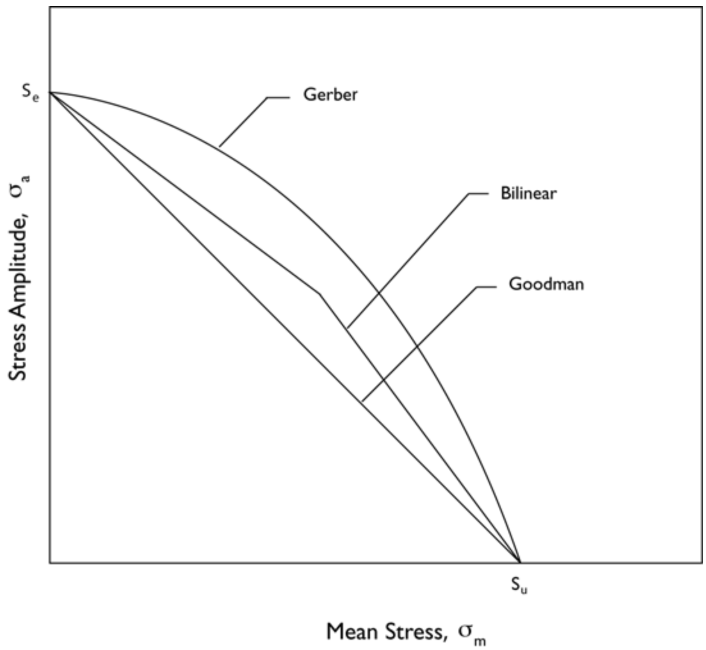


Figure 14-4: Simplified Haigh diagram.

There are a number of other factors that can affect the S-N curves:

- Environmental effects. A corrosive environment is negative for the fatigue life. A material usually having a fatigue limit might have none in a corrosive environment.
- Surface finish. Most data are obtained from polished specimens. Because microscopic irregularities are involved in the formation of fatigue cracks, a rough surface decreases fatigue life.
- Statistical size effects, as described earlier in this section.
- Residual stresses from manufacturing. This is sometimes used intentionally, for example, by shot peening that results in beneficial compressive stresses in the surface of the component.

LOW-CYCLE FATIGUE

Low-cycle fatigue is sometimes referred to as “strain based.” The reason is that the relevant parameter for describing LCF is strain rather than stress. Note though that physically it is the strain that does the damage in HCF as well. Because HCF occurs in the elastic regime, it is possible to use either stress or strain as the parameter, but the use of stress has historical and practical reasons.

The LCF analogy to the Basquin equation is the Coffin-Manson equation

$$\frac{\Delta\varepsilon_p}{2} = \varepsilon'_f(2N_f)^c \tag{14-6}$$

where $\Delta\varepsilon_p$ is the plastic strain range, and ε'_f and c are material parameters. The implication is that the plastic strain range is a straight line when plotted in a log-log S-N–type diagram against the number of cycles.

It is also possible to combine the Basquin and Coffin-Manson equations into a single expression, covering the entire range of LCF and HCF:

$$\frac{\Delta\varepsilon}{2} = \frac{\sigma'_f}{E}(2N_f)^b + \varepsilon'_f(2N_f)^c \tag{14-7}$$

The first term represents the elastic strain, the second the plastic strain, and $\Delta\varepsilon$ is the total strain range. The following table lists the parameters in Equation 14-7.

PARAMETER	DESCRIPTION
ε_f'	Fatigue ductility coefficient
c	Fatigue ductility exponent
σ_f'	Fatigue strength coefficient
b	Fatigue strength exponent

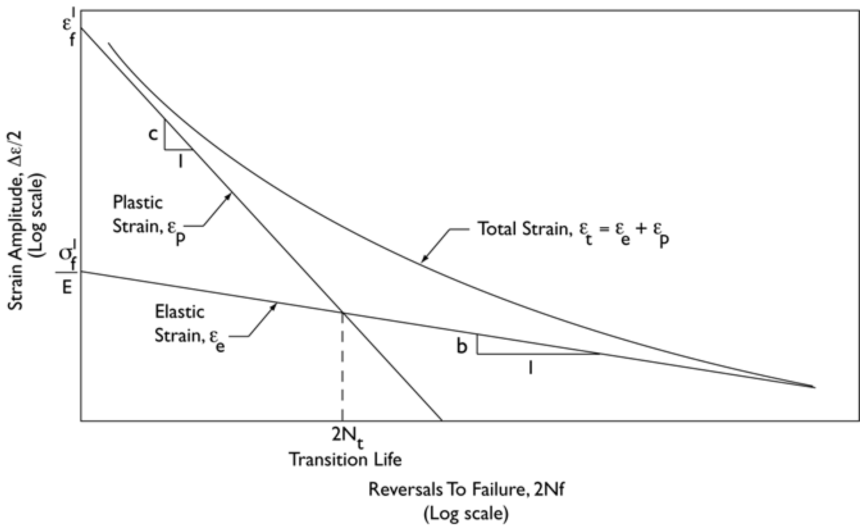


Figure 14-5: Strain-life curve.

As a criterion of the limit between LCF and HCF, you can use the transition life $2N_t$. It is the intersection between the lines formed by the Basquin and the Coffin-Manson curves.

Because plastic strains are important in LCF, analysis of such problems are more complex than the analysis of the corresponding HCF problem. The plastic strain must be obtained either from a full elasto-plastic analysis, or from some kind of extrapolation of an elastic analysis.

Loading Aspects

So far, the only load considered has constant amplitude, and the effect of it is a uniaxial stress.

In reality, the loads often have variable amplitude, and possibly also varying mean stress. The reason can be either that the service cycle contains several different well defined loadings, or that the load is random by its nature.

It is known that the order in which different loads are applied can have an effect, but most fatigue analyses ignore this fact due to the difficulties involved in such an analysis. Instead they treat the effect of each load as independent.

Assume that you have a set of stress cycles with amplitude/mean value pairs. Using the linear cumulative damage rule attributed to Palmgren and Miner, each such pair produces a relative damage

$$d_i = \frac{1}{N_i} \tag{14-8}$$

where N_i is the number of cycles to fatigue if only loads from pair i were acting.

In practice, all pairs with similar values are grouped together in classes ("bins"), represented by its class midpoint. If the number of cycles stored in bin i is n_i, the corresponding relative damage is

$$d_i = \frac{n_i}{N_i} \tag{14-9}$$

The limit for the possible fatigue life is then given by

$$\sum_i \frac{n_i}{N_i} = 1 \tag{14-10}$$

which implies infinite fatigue life for cases where this sum is less than 1.

For a random load, some type of cycle counting over a representative time interval must be used. This can be done from true measurements, or from a synthesized load history if the statistical properties of the load are known. The most commonly used method is called *rainflow counting* (Ref. 1), a procedure indicated in Figure 14-7. In the Structural Mechanics Module it is possible to perform a rainflow count on an arbitrary signal and split it into any number of amplitude/mean value pairs.

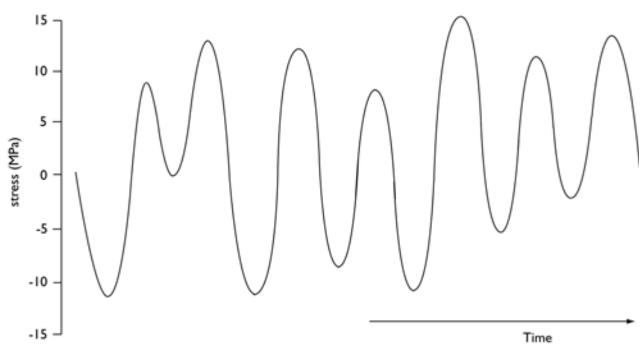


Figure 14-6: An example of a random load.

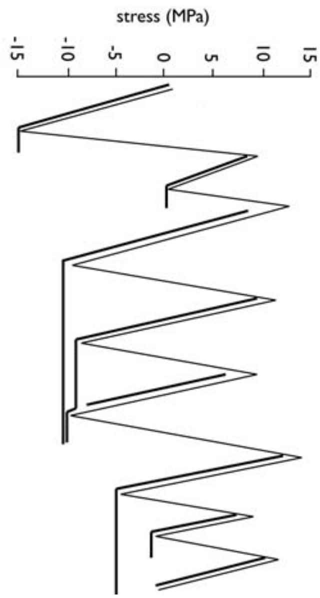


Figure 14-7: Rainflow counting of the cycles for the random load in the previous figure.

In real-life components the stress state is often multiaxial, in the sense that more than one principal stress is nonzero. Triaxial states of stress are unusual in this context, however, because fatigue cracks tend to appear on free surfaces, which by definition have a state of plane stress. The main exception is where contacting surfaces cause compression, so that fatigue cracks may instead develop below the surface.

If the directions of the principal stresses are fixed and the ratio between the principal stresses remain constant over the load history, the loading is said to be *proportional*. The problem is then just a matter of transforming uniaxial material data to biaxial material data.

In order for the opposite case, *nonproportional* loading, to occur, at least two independent loads must act on the structure. If the components of the stress tensor at the point of consideration do not only differ by a scale factor when the individual loads are applied, the loading is nonproportional. You find a simple case with nonproportional loading in the `shaft_with_fillet` model on page 346 in the *Structural Mechanics Module Model Library.*

If, however, the principal axes rotate between the loadings, the situation is much more complex. There are many methods suggested in the literature, both for LCF and HCF. A family of methods known as *critical plane* methods are popular, and are used in the Structural Mechanics Module, more information and details are found on page 374. In a critical plane method, some failure criterion is evaluated in all possible directions at a certain material point, and the maximum value is used.

Design Strategies

There are several possible strategies when designing components subjected to alternating loads.

1. Design for infinite lifetime by keeping the stresses sufficiently low. "Infinite" should here be interpreted as much longer than the service life of the component, so the method can be used also for a material without fatigue limit. This is the most common approach, and is preferred as long as it is not unfeasible.
2. Design for a fixed life (with sufficiently large factor of safety), after which the component is replaced.
3. Damage tolerant design, where the structure is inspected for cracks at regular intervals. In this case it is the growth rate of macroscopic cracks that sets the limit of the inspection interval, and fracture mechanics methods must be used in the analysis.

Summary

As a summary, it is necessary to answer the following questions when performing a fatigue analysis:

- Low-cycle or high-cycle fatigue?
- If low-cycle fatigue; will an elastic analysis be sufficient or is an elasto-plastic analysis required? An LCF analysis with both methods is shown in the `cylinder_with_hole` model on page 372 in the Structural Mechanics Module Model Library.
- Will the load have constant or variable amplitude? In the later case cumulative damage summing is required.
- Will the principal axes of the strain tensor be approximately constant or will they change between loadings?
- Are reliable material data available? How large is the scatter, and what risk of failure is acceptable?

Using the Structural Mechanics Module, it is possible to analyze both LCF and HCF problems. The procedure always starts with a stress analysis, and you then perform the evaluation against fatigue criteria as a separate postprocessing step. The fatigue analysis tools can handle the following types of loading:

- Proportional loading with constant amplitude
- Nonproportional loading with constant amplitude
- Proportional loading with nonconstant amplitude

Reference

1. "Standard practices for cycle counting in fatigue analysis," ASTM international, ASTM E 1049-85, 2005.

Further Reading

1. R.I. Stephens, A. Fatemi, R.R. Stephens, and H.O. Fuchs, *Metal Fatigue in Engineering*, Wiley-Interscience, 2000.

2. N.E. Dowling, *Mechanical Behavior of Materials* (3rd Edition), Prentice Hall, 2006.

3. D.F. Socie and G.B. Marquis, *Multiaxial Fatigue*, SAE, 1999.

How to Perform Fatigue Analysis

This section provides a detailed description of how to perform fatigue analysis using the Structural Mechanics Module. The theoretical background to the different fatigue models is also explained.

High-Cycle Fatigue

High-cycle fatigue typically means that the number of load cycles exceeds 10^4. The loading can be divided in nonproportional and proportional loading. The functions in the Structural Mechanics Module performing high cycle fatigue analysis are:

- `fatiguedamage` (see page 174 for details)
- `hcfmultiax` (see page 177 for details)

S-N CURVES (WÖHLER CURVES)

Fatigue data for high cycle fatigue are often given as S-N curves—often referred to as Wöhler curves—where the stress amplitude S (σ_a) is given as a function of the number of cycles to fatigue, N. Figure 14-8 shows a typical S-N curve. An S-N curve is experimentally determined.

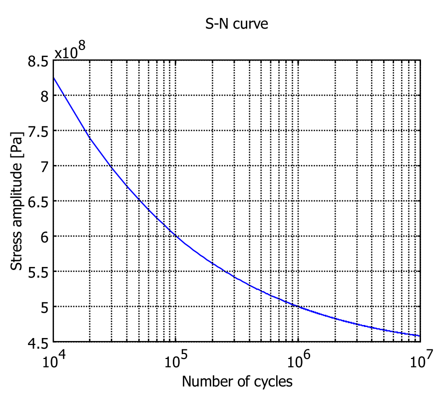


Figure 14-8: S-N curve for high-strength Iron Alloy UNS 4340.

The S-N curve depends on the mean stress, which can be characterized by the R value defined as

$$R = \frac{\sigma_{min}}{\sigma_{max}} \qquad (14\text{-}11)$$

Here σ_{min} is the minimum stress value and σ_{max} is the maximum stress value. The stress amplitude, σ_a, is defined as

$$\sigma_a = \frac{1}{2}(\sigma_{max} - \sigma_{min}) \qquad (14\text{-}12)$$

The most common fatigue tests are done for alternating loads ($R = -1$) and pulsating loads ($R = 0$). The allowable stress amplitude decreases with increasing R value.

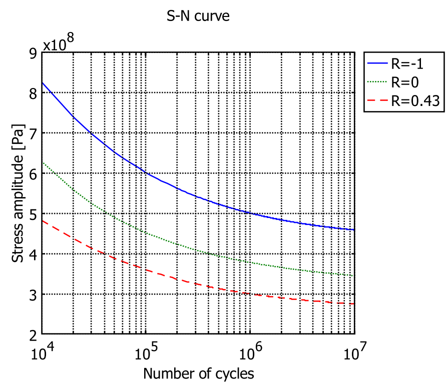


Figure 14-9: S-N curves for different R-values.

The function `fatiguedamage` requires the S-N data to be given as names of functions. The functions need to reside in a directory included in the COMSOL Script or MATLAB path. Input to the S-N curve is the number of cycles, output is the stress amplitude to fatigue for the given number of cycles. Examples of such a function is `sn_mat1_r_min1` included in the Structural Mechanics Module:

```
function stressAmp = sn_mat1_r_min1(n)
%SN_MAT1_R_MIN1 Compute stress amplitude to fatigue from number of cycles for
%               4340 (UNS G43400)
%               UTS 200 Ksi - 293K
%               R=-1; unnotched
%
%   STRESSAMP = SN_MAT1_R_MIN1(N) calculates the stress amplitude STRESSAMP
to fatigue
%   from the number of cycles N

%   Copyright (c) 1994-2007 by COMSOL AB

nExpr = 1;
```

```
exprs{1} = (6.134267E-02*log(n)^4 - 3.981468E+00*log(n)^3 +
1.000292E+02*log(n)^2 - 1.158936E+03*log(n) + 5.683208E+03)*1.000000e+006;
intervals(1) = 8.000000e+003;
intervals(nExpr+1) = 2.000000e+007;

n = n ;
if (n <= 8.000000e+003 )
  n = 8000;
  stressAmp = (6.134267E-02*log(n)^4 - 3.981468E+00*log(n)^3 +
1.000292E+02*log(n)^2 - 1.158936E+03*log(n) + 5.683208E+03)*1.000000e+006;
elseif (n >= 2.000000e+007);
  n = 20000000;
  stressAmp = (6.134267E-02*log(n)^4 - 3.981468E+00*log(n)^3 +
1.000292E+02*log(n)^2 - 1.158936E+03*log(n) + 5.683208E+03)*1.000000e+006;
else
  if (n > 8.000000e+003 & n <= 2.000000e+007)
    stressAmp = (6.134267E-02*log(n)^4 - 3.981468E+00*log(n)^3 +
1.000292E+02*log(n)^2 - 1.158936E+03*log(n) + 5.683208E+03)*1.000000e+006;
  end
end
r = -1.000000e+000;
stressAmp = stressAmp*(1-r)/2;
```

The fatigue data in the example above is extracted from the COMSOL Material Library, which is an add-on product to COMSOL Multiphysics. This is done using the `matlibfatigue` function. For details see `matlibfatigue` on page 184 in the *Structural Mechanics Module Reference Guide*.

Note: The S-N curves in the Material Library are given on the form σ_{max} as function of number of cycles, but the fatigue functions in the Structural Mechanics Module requires σ_a as function of the number of cycles. The transformation is automatically handled by the `matlibfatigue` function.

Using Equation 14-11 and Equation 14-12, σ_a can be calculated from σ_{max} as

$$\sigma_a = \sigma_{max}\frac{(1-R)}{2} \tag{14-13}$$

For R values between two specified S-N curves, the values are calculated by interpolating between the two S-N curves. You pass the R values for the specified S-N functions to the `fatiguedamage` function using the `rvalue` property.

For R values outside the available S-N curves a mean stress correction is calculated by interpolating the value for $R = -1$ and then using the mean stress correction method specified through the `method` property to the `fatiguedamage` function. The same applies if you only have the S-N curve for $R = -1$. There are different mean stress correction theories to use if you only have fatigue data for $R = -1$, that is, an alternating load. The Structural Mechanics Module supports two such methods: Gerber and Goodman. The methods are based on the ratio between mean stress and the ultimate stress σ_{uts}. Figure 14-10 compares the stress amplitude as function of the mean stress for the two methods.

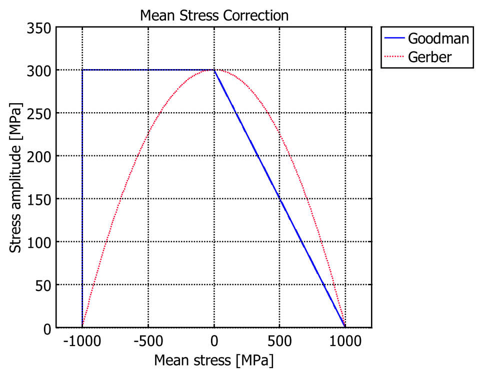


Figure 14-10: Mean stress compensation for the Gerber and Goodman methods for a material with an ultimate stress of 1000 MPa and an endurance limit of 300 MPa for alternating loads.

You specify the ultimate stress to the `fatiguedamage` function using the `params` property.

NONPROPORTIONAL LOADING, CONSTANT AMPLITUDE

Use the function `hcfmultiax` for this type of analysis. The properties in the following section refer to the `hcfmultiax` function.

Theory Background

Nonproportional loading is defined as any state of time-varying stress in which the orientations of the principal stress axes change with respect to axes that are fixed with respect to the component. A simple example of nonproportional loading is a shaft exposed to both cyclic bending and torsion.

The critical plane is defined as the plane where the fatigue crack occurs. Different models use different criteria to determine the critical plane. A successful model must be able to predict both the fatigue life and the dominant failure plane. For nonproportional loading you need to examine the loading history for all possible planes (directions) in order to find the critical plane. A critical plane model evaluates the stresses on different planes (directions) in the material and maximizes some type of damage criterion with respect to all possible directions. You control the resolution of the direction search with the property `anglestep`. `anglestep` is the step in angle subdividing the longitude of the unit sphere and controlling the subdivision in the latitude direction. This means that the number of searches are inversely proportional to the square of the step. The critical plane model in the Structural Mechanics Module is the Findley criterion. It can be stated as

$$\left(\frac{\Delta\tau}{2}+k\cdot\sigma_n\right)_{\max} = f \tag{14-14}$$

where k and f are material parameters. You pass these material parameters to the `hcfmultiax` function through the property `params`. The Findley criterion states stress combinations giving the fatigue limit. In Equation 14-14, $\Delta\tau$ is the maximum shear stress range of the cycle, and σ_n is the maximum normal stress during the cycle. The left-hand side of Equation 14-14 must for each material point (for example, a node in an FE analysis) be computed for a large number of directions until the maximum value is found. The fatigue usage factor fus is the ratio between the Findley criterion and the material parameter f. A value below 1 means that the component is loaded below the fatigue limit.

$$\mathrm{fus} = \frac{\left(\frac{\Delta\tau}{2}+k\cdot\sigma_n\right)_{\max}}{f} \tag{14-15}$$

On a given plane the normal stress is a scalar, but the shear stress is a two-dimensional vector. This requires an interpretation of $\Delta\tau$. The most strict interpretation is that $\Delta\tau$ is the diameter of the smallest circle inscribing the path that the τ vector creates during a load cycle. This calculation is nontrivial, however, and takes significant computer resources. An alternative is to use the maximum distance between any two points on the path instead. This simplified procedure could in extreme cases underestimate $\Delta\tau$ by 13%, but in most cases the result is much better. The summation in Equation 14-14 further reduces the error. The `hcfmultiax` function supports both these methods for calculating $\Delta\tau$ through the property `opt`.

To find the two material parameters k and f you need two fatigue tests with different loading conditions. This can, for example, be pure tension and pure torsion, but there are other possibilities. For axial loading, the following relation is valid:

$$\sqrt{\left(\frac{\sigma_{max} - \sigma_{min}}{2}\right)^2 + (k \cdot \sigma_{max})^2} + k \cdot \sigma_{max} = 2f \tag{14-16}$$

Here σ_{max}, and σ_{min} are the maximum and minimum stresses at the fatigue limit, that is, infinite life. In a pure (fully reversed) torsion test with an amplitude τ_a of the torsional shear stress, the corresponding relation is

$$\tau_a = \frac{f}{\sqrt{1 + k^2}} \tag{14-17}$$

If only uniaxial test data with a single R value is available, it is possible to estimate k from the ratio between the fatigue limits under different conditions for a similar material.

Conducting a Fatigue Analysis

An analysis of high-cycle fatigue with nonproportional loading consists of the following steps:

1. Perform a finite element analysis (FEA) for the basic load cases.
2. Calculate all stress components from the FEA model for the different basic load cases at the locations where you are interested to find the fatigue damage. You do this by calling `posteval` from the command prompt.
3. Define the loading history by combining the basic load cases.
4. Find appropriate material data in the form of Findley parameters, k and f.
5. Calculate the fatigue usage factor using the `hcfmultiax` function. You find a detailed description in the entry for `hcfmultiax` on page 177 in the *Structural Mechanics Module Reference Guide*.
6. Plot the fatigue damage and look at the stress history. You can plot the fatigue damage using the `postdataplot` function.

You find an example of a high-cycle fatigue analysis with nonproportional loading in the model “Shaft with Fillet” on page 346.

PROPORTIONAL LOADING, NONCONSTANT AMPLITUDE

You use the function `fatiguedamage` for this type of analysis. The properties in the following section refer to the `fatiguedamage` function.

When the loading history is not deterministic, the question of how to characterize the load cycles from a fatigue point of view arises. An example of such a load (or stress) history is shown in Figure 14-11.

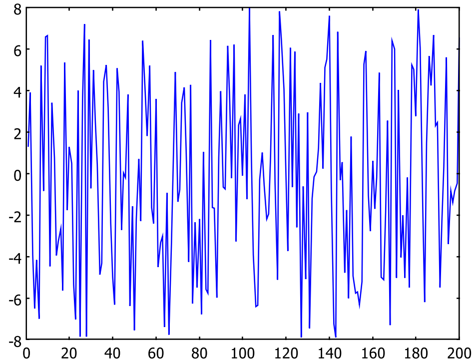


Figure 14-11: Example of a nondeterministic load history.

There are several methods for determining cycles having different ranges and possibly corresponding mean values. One commonly used method is “Rainflow counting” as described in Ref. 1.

If fatigue data, S-N curves are available it is possible to determine the allowable number N_i of cycles to fatigue for each such pair i consisting of a mean and amplitude stress. Using the linear cumulative damage rule attributed to Palmgren and Miner, each such pair would produce a relative damage

$$d_i = \frac{1}{N_i} \tag{14-18}$$

In practice, all pairs with similar values of the mean stress and amplitude are grouped together in classes (“bins”), represented by its class midpoint. If the number of cycles stored in bin i is n_i, the corresponding relative damage is

$$d_i = \frac{n_i}{N_i} \tag{14-19}$$

The limit for the possible fatigue life is then given by

$$\sum_i \frac{n_i}{N_i} = 1 \tag{14-20}$$

In the Structural Mechanics Module there is a function named `rainflow`, which performs a rainflow count on an arbitrary signal and returns the number of occurrences in a matrix of (range, mean) bins.

This type of counting is useful only if the principal stresses are not rotating; in practice this often means that a single load controls the stress history, or that the critical point has a uniaxial stress state. It is theoretically possible to include effects of multiaxiality, but usually a single stress component (for example, the largest principal stress) is used.

The total fatigue damage factor damtot calculated by the Structural Mechanics Module function `fatiguedamage` is defined as

$$\mathtt{damtot} = \sum_i \frac{n_i}{N_i} \qquad (14\text{-}21)$$

Conducting a Fatigue Analysis

An analysis of high-cycle fatigue with nonconstant amplitude and proportional loading consists of the following steps:

1. Perform a finite element analysis for a unit loading factor.
2. Calculate all stress components from your FEA model at the locations where you are interested to find the total fatigue damage factor damtot. You do this by calling the `posteval` function.
3. Define the loading history as load factors either from measurements or from statistical methods.
4. Perform a Rainflow count of the loading history using the `rainflow` function. This results in a binning of your load. You can plot the result of the count using the `stem` plot function.
5. Find appropriate material data for different mean stresses in form of S-N curves and write functions (M-files) returning the stress amplitude giving the number of cycles or use a single S-N curve for $R = -1$ and specify a mean stress correction method through the `method` property.
6. Calculate the total fatigue damage factor damtot and the damage distribution using the `fatiguedamage` function. You find a detailed description of `fatiguedamage` on page 174 in the *Structural Mechanics Module Reference Guide*.
7. Plot the total fatigue damage factor damtot and the damage distribution.

You find an example of a high-cycle fatigue analysis with proportional loading in the model "Frame with Cutout" on page 357.

Low-Cycle Fatigue

When studying low-cycle fatigue, it has been found that it is more relevant to use strain as the important parameter in the fatigue laws. There is always significant cyclic plastic deformation involved in low-cycle fatigue situations. The functions in the Structural Mechanics Module performing low-cycle fatigue analysis are:

- `lcfmultiaxlin` (see page 179 in the *Structural Mechanics Module Reference Guide* for details)
- `lcfmultiaxpla` (see page 181 in the *Structural Mechanics Module Reference Guide* for details)

For a uniaxial case, it is often possible to use an expression of the type

$$\frac{\Delta\varepsilon}{2} = \frac{\sigma_f'}{E}(2N_f)^b + \varepsilon_f'(2N_f)^c \tag{14-22}$$

Here $\Delta\varepsilon$ is the total strain range, and the two terms on the right-hand side represent the elastic and the plastic strain contributions. $2N_f$ is the number of load reversals to fatigue (so that N_f is the number of full cycles). In addition to the modulus of elasticity, E, there are four independent parameters: two coefficients, and two exponents.

PARAMETER	DESCRIPTION
ε_f'	Fatigue ductility coefficient
c	Fatigue ductility exponent
σ_f'	Fatigue strength coefficient
b	Fatigue strength exponent

There are many related models, considering, for example, mean stress effects, differences between shear and tension, or multiaxiality. One popular such model is the Smith-Watson-Topper (SWT) model. This is a type of critical plane model, where the plane normal to the maximum principal strain range is considered.

$$\sigma_{n,\max}\cdot\frac{\Delta\varepsilon_1}{2} = \frac{\sigma_f'^2}{E}(2N_f)^{2b} + \sigma_f'\varepsilon_f'(2N_f)^{b+c} \tag{14-23}$$

The left-hand side is commonly called the SWT parameter and contains the maximum normal stress during the cycle on the used plane. The material parameters in Equation 14-22 and Equation 14-23 are the same. You specify these parameters to the functions `lcfmultiaxlin` and `lcfmultiaxpla` through the property `params`.

In general, the stress and strain in Equation 14-22 and Equation 14-23 must be computed using plasticity theory. There are two possible approaches. The first is to compute a complete cycle, using a full elasto-plastic analysis. In this type of analysis, it is important to model the cyclic plastic behavior of the material appropriately. This means that kinematic hardening models are more suitable than isotropic models. It might also be necessary to analyze more than one load cycle in order to obtain a stabilized stress-strain cycle.

The second case occurs if the highly stressed region is localized. In this common case, it is possible to determine the stress and strain range using an elastic analysis and then externally compute an approximation to the plastic stresses and strains. Neuber's rule states that for a notch, the product of elastically computed stress and strain is equal to the product of the actual, inelastic, stress and strain. Strictly speaking, it is defined in terms of a uniaxial stress state. In practice, the stress states are often multiaxial, so here Neuber's rule is expressed in equivalent stresses and strains.

$$ {}^{e}\sigma^{eq} \cdot {}^{e}\varepsilon^{eq} = \sigma^{eq} \cdot \varepsilon^{eq} \tag{14-24} $$

In Equation 14-24 the left side has an "e" denoting the results of an elastic analysis, while the right side contains the actual values.

In strain-based fatigue analysis it is customary to assume an Ramberg-Osgood material law when modeling the cyclic plastic behavior

$$ \varepsilon = \frac{\sigma}{E} + \left(\frac{\sigma}{K}\right)^{\frac{1}{n}} \tag{14-25} $$

The parameters in Equation 14-25 must be the cyclic values and not the monotonic values obtained from a standard tensile test.

Hoffman and Seeger has developed an algorithm for approximate computation of the stress and strain amplitude in a multiaxial case. You specify the material properties to the function `lcfmutiaxlin` through the property `params`. Initially, Equation 14-24 and Equation 14-25 are solved together for obtaining the true equivalent (in von Mises sense) stresses and strains. It is then possible to approximate the major principal strain and corresponding stress as

$$ \varepsilon_1 = \frac{1 - \bar{\nu} a}{\sqrt{1 - a + a^2}} \cdot \varepsilon^{eq} \tag{14-26} $$

$$\sigma_1 = \frac{1}{\sqrt{1-a+a^2}} \cdot \sigma^{eq} \tag{14-27}$$

These two relations contain the parameters a and $\bar{\nu}$. The latter is an effective Poisson's ratio defined as

$$\bar{\nu} = \frac{1}{2} - \left(\frac{1}{2} - \nu\right) \cdot \frac{\sigma^{eq}}{E\varepsilon^{eq}} \tag{14-28}$$

A biaxiality factor, ϕ, is computed as the ratio between the two in-plane elastic principal strains:

$$\phi = \frac{{}^{e}\varepsilon_2}{{}^{e}\varepsilon_1} \tag{14-29}$$

The parameter a is then defined as

$$a = \frac{\phi + \bar{\nu}}{1 + \bar{\nu}\phi} \tag{14-30}$$

The stress σ_1 and the strain ε_1 can now be used for computing the fatigue life using, for example, the SWT equation above.

CONDUCTING A FATIGUE ANALYSIS

Depending of whether the highly loaded area can be considered localized or not, two different methods can be used.

General Case

An analysis of low cycle fatigue with nonlocalized stresses consists of the following steps:

1. Perform a full elasto-plastic FEA analysis for as many cycles as needed to get a stabilized stress field.
2. Calculate all stress and strain components during a complete cycle at the locations where you are interested to find the fatigue damage. Do this by calling the `posteval` function.
3. Find appropriate material data for the Smith-Watson-Topper (SWT) fatigue model.
4. Calculate the fatigue damage using the `lcfmultiaxpla` function. You find a detailed description in the entry for `lcfmultiaxpla` on page 181 in the *Structural Mechanics Module Reference Guide*.

5 Plot the fatigue damage by calling the `postdataplot` function.

You find an example of a low-cycle fatigue analysis using a full elasto-plastic analysis in the model "Cylinder with Hole" on page 372.

Simplified Analysis

An analysis of low cycle fatigue with localized stresses consists of the following steps:

1. Perform a linear elastic FEA analysis to get the stress field.
2. Calculate the principal stress components at the locations where you are interested to find the fatigue damage. Do this calculation by calling the `posteval` function.
3. Find appropriate material data for the Smith-Watson-Topper (SWT) fatigue model and your simplified linear elastic to elasto-plastic calculation.
4. Calculate the fatigue damage using the `lcfmultiaxlin` function. You find a detailed description in the entry for `lcfmultiaxlin` on page 179 in the *Structural Mechanics Module Reference Guide*.
5. Plot the fatigue damage by calling the `postdataplot` function.

You find an example of a low-cycle fatigue analysis using a linear elastic analysis in the model "Cylinder with Hole" on page 372.

References

1. "Standard practices for cycle counting in fatigue analysis," ASTM international, ASTM E 1049-85, 2005.

2. D.F. Socie and G.B. Marquis, *Multiaxial Fatigue*, SAE, ISBN 0-7680-0453-5.

15

Glossary

This glossary contains finite element modeling terms in a structural mechanics context. For mathematical terms, and geometry and CAD terms specific to the COMSOL Multiphysics software and documentation see the glossary in the *COMSOL Multiphysics User's Guide*. For references to more information about a term, see the index.

Glossary of Terms

anisotropy Variation of material properties with direction. Both global and local user defined coordinate systems can be used to define anisotropic material properties.

augmented Lagrangian method Segregated solution method, where the solver switches between solving for the contact pressure and the displacements, used when modeling contact.

axial symmetry Symmetry in both load and geometry, solves for the radial (r) and axial (z) displacement.

bar A line element that only has translational degrees of freedom, capable of sustaining axial forces, with no bending moments, torsional moments, or shear forces. Can be used on lines in 1D, 2D, and 3D.

beam A line element having both translational and rotational degrees of freedom. Capable of sustaining axial forces, bending moments, torsional moments, and shear forces. Can be used on curves in 2D and 3D.

benchmark Standard test designed to evaluate the accuracy or efficiency of a finite element system or model.

body forces Forces distributed through the volume of a body.

buckling The sudden collapse or reduction in stiffness of a structure under a critical combination of applied loads.

cable A tension-only truss member used to model large deformation including sag.

Cauchy stress The most fundamental stress measure defined as force/deformed area in fixed directions not following the body.

compliance matrix The inverse of the elasticity matrix. See *elasticity matrix.*

constitutive equations The equations formulating the stress-strain relationship of a material.

constraint Constrains the displacement or rotations to zero or a specified value.

contact model The mathematical method to model bodies that come into contact with each other.

contact pair A contact pair consists of some slave and master boundaries and is used for contact modeling.

continuum application modes The application modes that solve for the displacement field without involving rotations. Solid, Stress-Strain; Plane Stress; Plane Strain; and Axial Symmetry, Stress-Strain are the continuum application modes.

coordinate system Global Cartesian, local geometrical, application specific, and user defined coordinate systems. Loads, constraints, material properties, and postprocessing variables are defined in a specific coordinate system.

creep Time-dependent material nonlinearity that usually occurs in metals at high temperatures in which the effect of the variation of stress and strain with time is of interest.

damping Dissipation of energy in a vibrating structure. A common assumption is viscous damping where the damping is proportional to the velocity. See also *Rayleigh damping*.

eigenfrequency analysis Solving for the undamped natural frequencies and vibration modes of a structure.

elasticity matrix The matrix D relating strain to stresses:

$$\sigma = D\varepsilon$$

elasto-plastic material A material model where the material exhibits both elastic and plastic behavior. See also *plasticity*.

equilibrium equation The equation expressing the equilibrium formulated in the *stress* components.

fatigue A term describing the phenomena where a component fails after repeated loadings and unloadings.

first Piola-Kirchhoff stress A rather mathematical stress measure used in the hyperelastic material model, its conjugate strain is the displacement gradient.

flexibility matrix The inverse of the *elasticity matrix*. See *elasticity matrix*.

free vibration The undamped vibration of a structure after it is displaced from the equilibrium position and released. See also *eigenfrequency analysis*.

frequency response analysis A harmonic analysis solving for the steady-state response from a harmonic excitation. Typically a frequency sweep is performed, solving for many excitation frequencies at one time.

geometric nonlinearity See *large deformation*.

Green-Lagrange strain Nonlinear strain measure used in large-deformation analysis. In a small strain, large rotation analysis, the *Green-Lagrange* strain corresponds to the engineering strain, with the strain values interpreted in the original directions. The *Green-Lagrange* strain is a natural choice when formulating a problem in the undeformed state. The conjugate stress is the *second Piola-Kirchhoff stress*.

hyperelastic material Material where the stresses are computed from a strain energy density function. Often used to model rubber.

initial strain The strain in a stress-free structure before it is loaded.

initial stress The stress in a non-deformed structure before it is loaded.

isotropic material A material where the material properties are independent of direction.

isotropic hardening A hardening model for an *elasto-plastic material* where the yield surface increases in size but maintains its original shape.

kinematic hardening A hardening model for an *elasto-plastic material* where the yield surface is translated to a new position in the stress space as the plastic strain is increased, with no change in size or shape.

large deformation The deformations are so large so the nonlinear effect of the change in geometry or stress stiffening need to be accounted for.

linear buckling analysis Solves for the linear buckling load using the eigenvalue solver.

mass damping parameter Rayleigh damping parameter, the coefficient in front of the mass matrix.

master boundary One side of a contact pair, the slave boundary is prohibited to penetrate the master boundary.

Mindlin plate A thick plate including shear deformation. See also *plate*.

mixed formulation A formulation where the pressure have been added as a dependent variable, used for nearly incompressible materials to avoid numerical problems.

Mooney-Rivlin material A hyperelastic material model with three model parameters, the model is based on modified strain invariants.

Neo-Hookean material model A hyperelastic material model with two model parameters, the model is based on modified strain invariants.

nonlinear geometry See *large deformations*.

orthotropic material An orthotropic material has at least two orthogonal planes of symmetry, where material properties are independent of direction within each plane. Such materials require nine independent variables (that is, elastic constants) in the constitutive equations.

parametric analysis An analysis which finds the solution dependence due to the variation of a specific parameter.

pinned A constraint condition where the displacement degrees of freedom are fixed but the rotational degrees of freedom are free, typically used for frames modeled using beams and truss elements.

plane strain An assumption on the strain field where all out-of-plane strain components are assumed to be zero.

plane stress An assumption on the stress field, all out-of-plane stress components are assumed to be zero.

plasticity A time-independent material nonlinearity. Three classes of plastic behavior are considered: *perfectly plastic*, *isotropic hardening*, *kinematic hardening*.

plate Thin structure loaded in the normal direction.

primary creep The initial creep stage where the strain rate is decreasing with time.

principle of virtual work States that the variation in internal strain energy is equal to the work done by external forces.

principal stresses/strains Normal stresses/strains with no shear components that act on the principal planes. The magnitude of the principal stresses/strains are independent of the coordinate system used.

quasi-static transient analysis The loads vary slowly so inertia terms can be neglected. A transient thermal analysis coupled with a structural analysis can often be treated as quasi-static.

Rayleigh damping A viscous damping model where the damping is proportional to the mass and stiffness, through the mass and stiffness damping parameters.

rotational degrees of freedom Degrees of freedom associated with a rotation around an axis. Beams, Mindlin plates, and shells have *rotational degrees of freedom*.

secondary creep A creep regime where the strain rate is almost constant.

second Piola-Kirchhoff stress Conjugate stress to *Green-Lagrange strain* used in large deformation analysis.

shell elements A thin element where both bending and membrane effects are included.

slave boundary One side of a contact pair, the slave boundary is prohibited to penetrate the master boundary.

static analysis An analysis where the loads and constraints are constant in time.

strain Relative change in length, a fundamental concept in structural mechanics.

stress Internal forces in the material, normal stresses are defined as forces/area normal to a plane, and shear stresses are defined as forces/area in the plane. A fundamental concept in structural mechanics.

stiffness damping parameter Rayleigh damping parameter, the coefficient in front of the stiffness matrix.

strain energy The energy stored by a structure as it deforms under load.

transient analysis A time-dependent analysis, taking into account mass, mass moment of inertia, and damping.

tertiary creep The creep stage where the strain rate increases very rapidly, followed by eventual failure.

Tresca stress An effective stress measure that is equal to the maximum shear stress.

truss See *bar.*

viscoelasticity A time-dependent material nonlinearity. Viscoelastic materials have a time-dependent response, even if the loading is constant. Many polymers and biological tissues exhibit such a behavior. *Linear viscoelasticity* is a commonly used approximation where the stress depends linearly on the strain and its time derivatives.

INDEX