COMSOL MULTIPHYSICS

STRUCTURAL MECHANICS MODULE

MODEL LIBRARY

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 Model Library

Version: October 2007 COMSOL 3.4

CONTENTS

Chapter 1: Introduction

Chapter 2: Acoustic-Structure Interaction

Chapter 3: Automotive Application Models

Chapter 4: Benchmark Models

Chapter 5: Bioengineering

Chapter 6: Civil Engineering Models

Chapter 7: Contact and Friction Models

Chapter 8: Dynamics and Vibration Models

Chapter 9: Fatigue Models

Chapter 11: Fracture Models

Chapter 12: Nonlinear Material Models

Chapter 15: Thermal-Structure Interaction

1

Introduction

The *Structural Mechanics Module Model Library* consists of a set of models that simulate problems in various areas of structural mechanics and solid mechanics engineering. Their purpose is to assist you in learning, by example, how to model sophisticated structural elements and systems. Through them you can tap the expertise of the top researchers in the field, examining how they approach some of the most difficult modeling problems you might encounter. You can thus get a feel for the power that COMSOL Multiphysics offers as a modeling tool. In addition to serving as a reference, the models can also give you a big head start if you are developing a model of a similar nature.

We have divided these models into application areas such as automotive applications, civil engineering, fluid-structure interaction, and piezoelectric effects. The models illustrate the use of the various structural-mechanics specific application modes from which we built them. These specialized application modes are not available in the base COMSOL Multiphysics package, and they come with their own graphical user interfaces that make it quick and easy to access their power. You can even modify them for custom requirements. COMSOL Multiphysics itself is very powerful and, with sufficient expertise in a given field, you certainly could develop these application modes by yourself—but why spend the hundreds or thousands of hours that would be necessary when our team of experts has already done the work for you?

Note that the model descriptions in this book do not contain details on how to carry out every step in the modeling process. Before tackling these in-depth models, we urge you to first read the second book in the Structural Mechanics Module documentation set. Titled the *Structural Mechanics Module User's Guide*, it introduces you to the basic functionality in the module, reviews new features in the version 3.4 release, covers basic modeling techniques with tutorial and benchmark example models, and includes reference material of interest to those working in structural mechanics. A third book, the *Structural Mechanics Module Reference Guide*, contains reference material about command-line programming and functions. It is available in HTML and PDF format from the COMSOL Help Desk. For more information on how to work with the COMSOL Multiphysics graphical user interface, please refer to the *COMSOL Multiphysics User's Guide* or the *COMSOL Multiphysics Quick Start* manual. An explanation on how to model with a programming language is available in yet another book, the *COMSOL Multiphysics Scripting Guide*.

The book in your hands, the *Structural Mechanics Module Model Library*, provides details about a large number of ready-to-run models that illustrate real-world uses of the module. Each entry comes with theoretical background as well as instructions that illustrate how to set it up. They were written by our staff engineers who have years of experience in structural mechanics; they are your peers, using the language and terminology needed to get across the sophisticated concepts in these advanced topics.

Finally note that we supply these models as COMSOL Multiphysics model files so you can open them in COMSOL Multiphysics for immediate access, allowing you to follow along with these examples every step along the way.

Model Library Guide

The table below summarizes key information about the entries in this model library. The solution time is the elapsed time measured on a machine running Windows Vista with a 2.6 GHz AMD Athlon X2 Dual Core 500 CPU and 2 GB of RAM. For models with a sequential solution strategy, the Solution Time column shows the elapsed time for the longest solution step. The Application Mode column contains the application modes (such as Plane Stress) we used to solve the model. The following columns indicate the analysis type (such as eigenfrequency), and if the model includes parametric studies, buckling, elasto-plastic materials, or multiphysics couplings.

MODEL	PAGE	SOLUTION TIME	APPLICATION MODE	STATIC	EIGENFREQUENCY	TIME DEPENDENT	FREQUENCY RESPONSE	PARAMETRIC	QUASI-STATIC TRANSIENT	LINEAR BUCKLING	ELASTO-PLASTIC MATERIAL	FATIGUE	MULTIPHYSICS
ACOUSTIC-STRUCTURE INTERACTION													
Vibrations of a disk backed by an air-filled cylinder	10	8 s	Mindlin Plate, Acoustics		√								√
Acoustic-structure interaction	22	2 min	Solid, Stress-Strain; Acoustics				√						√
Piezoacoustic transducer	34	1 s	Piezo Plane Strain; Acoustics				√						√
AUTOMOTIVE APPLICATIONS													
Diesel engine piston	46	2 min	Solid, Stress-Strain; Heat Transfer by Conduction	√									√
Fuel cell bipolar plate	62	20 s	Solid, Stress-Strain; Heat Transfer by Conduction	√									√
Spinning gear	82	8 s	Plane Stress	√				√					
Automobile wheel rim	108	13 min	Solid, Stress-Strain	√				√					
BENCHMARK MODELS													
Wrapped cylinder	122	1 s	Solid, Stress-Strain	√									
Large deformation beam	134	29 s	Plane Stress	√				√		√			
Thick plate	142	6 s	Solid, Stress-Strain	√									
Kirsch plate	152	1 s	Plane Stress	√									
Thick wall cylinder	161	2 s	Plane Strain	√									√
In-plane frame	169	1 s	In-Plane Euler Beam		√								
Thermally loaded beam	177	1 s	3D Euler Beam	√									√
In-plane truss	186	1 s	In-Plane Truss	√									
Scordelis-Lo roof	197	1 s	Shell	√									

MODEL	PAGE	SOLUTION TIME	APPLICATION MODE	STATIC	EIGENFREQUENCY	TIME DEPENDENT	FREQUENCY RESPONSE	PARAMETRIC	QUASI-STATIC TRANSIENT	LINEAR BUCKLING	ELASTO-PLASTIC MATERIAL	FATIGUE	MULTIPHYSICS
BIOENGINEERING													
Blood vessel	210	9 min	Solid, Stress-Strain; Navier-Stokes			√			√				√
Bladder	222	14 min	Solid, Stress-Strain			√							
Biomedical stent	235	138 min	Solid, Stress-Strain	√				√			√		
CIVIL ENGINEERING													
Pratt truss bridge	246	4 s	3D Euler Beam, Shell	√	√								
CONTACT AND FRICTION													
Sliding wedge	262	53 s	Plane Stress	√									
Cylinder roller contact	271	17 s	Plane Strain	√				√					
Tube connection	283	17 min	Solid, Stress-Strain	√				√					
Snap hook fastener	297	38 min	Solid, Stress-Strain	√				√			√		
Spherical punch	310	106 min	Axial Symmetry Stress-Strain	√				√			√		
DYNAMICS AND VIBRATION													
Rotor	320	8 s	Solid, Stress-Strain		√								
Free cylinder	326	1 s	Axial Symmetry Stress-Strain		√								
Harmonically excited plate	334	25 s	Mindlin Plate				√						
FATIGUE**													
Shaft with fillet	346**	19 s	Solid, Stress-Strain	√				√				√	
Frame with cutout	357**	5 s	Shell	√				√				√	
Cylinder with hole	372**	10 min	Solid, Stress-Strain	√				√			√	√	
Fatigue analysis of an automobile wheel rim	386**	30 min	Solid, Stress-Strain	√				√				√	
FLUID-STRUCTURE INTERACTION													

MODEL	PAGE	SOLUTION TIME	APPLICATION MODE	STATIC	EIGENFREQUENCY	TIME DEPENDENT	FREQUENCY RESPONSE	PARAMETRIC	QUASI-STATIC TRANSIENT	LINEAR BUCKLING	ELASTO-PLASTIC MATERIAL	FATIGUE	MULTIPHYSICS
Freezing soil	404	5 min	Heat Transfer, Axial Symmetry Stress-Strain, Darcy's Law			√							√
Obstacle in fluid	417	11 min	Solid, Stress-Strain; Incompressible Navier-Stokes; Moving Mesh (ALE)			√							√
FRACTURE													
Single edge crack	426	1 s	Plane Stress	√									
NONLINEAR MATERIAL MODELS													
Elasto-plastic plate	438	42 s	Plane Stress	√				√			√		
Viscoelastic material	443	50 s	Plane Strain, PDE	√					√				√
Hyperelastic seal	467	4 min	Plane Strain	√				√					√
Thermally induced creep	479	2 min	Axial Symmetry Stress-Strain, PDE						√				√
Traffic tunnel	500	61 min	Plane Strain	√									
Flexible and smooth strip footing	516	43 s	Plane Strain	√				√			√		
PIEZOELECTRIC EFFECTS													
Piezoceramic tube	528	1 s	Piezo Axial Symmetry	√									√
A piezoelectric shear actuated beam	541	7 s	Piezo Solid; Solid, Stress-Strain	√									√
Composite piezoelectric transducer	553	6 min	Piezo Solid; Solid, Stress-Strain		√		√						√
Piezoresistive elevator button	565	11 min	Solid, Stress-Strain; Electric Currents***						√				√
SAW gas sensor	582	11 s	Piezo Plane Strain		√								√
STRESS-OPTICAL EFFECTS													

MODEL	**PAGE**	**SOLUTION TIME**	**APPLICATION MODE**	**STATIC**	**EIGENFREQUENCY**	**TIME DEPENDENT**	**FREQUENCY RESPONSE**	**PARAMETRIC**	**QUASI-STATIC TRANSIENT**	**LINEAR BUCKLING**	**ELASTO-PLASTIC MATERIAL**	**FATIGUE**	**MULTIPHYSICS**
Stress-optical effects in a silica-on-silicon waveguide	598	2 s	Plane Strain, Perpendicular Hybrid-Mode Waves	√	√								√
Stress-optical effects with generalized plane strain	615	7 s	Plane Strain, Perpendicular Hybrid-Mode Waves	√	√								√
THERMAL-STRUCTURAL INTERACTION													
Thermal stresses in a layered plate	634	1 s	Plane Stress, Heat Transfer by Conduction	√									√
Surface mount resistor	644	3 min	General Heat Transfer; Solid, Stress-Strain	√	√								√
Heating circuit	657	2 min	Solid, Stress-Strain; General Heat Transfer; Thin Conductive Shell; Shell	√									√
Microrobot 3D	673	10 min	General Heat Transfer; Solid, Stress-Strain; Shell, Conductive Media DC			√		√					√
Thermal expansion in a MEMS device using the Material Library	689	19 s	Solid, Stress-Strain; Heat Transfer by Conduction	√									√
Heat generation in a vibrating structure	703	31 s	Solid, Stress-Strain; Heat Transfer by Conduction	√			√						√
TUTORIAL MODELS													
Component static	17*	1 s	Plane Stress	√									
Component eigen	28*	1 s	Plane Stress		√								
Component transient	32*	17 s	Plane Stress			√							
Component frequency	41*	22 s	Plane Stress				√						

MODEL	PAGE	SOLUTION TIME	APPLICATION MODE	STATIC	EIGENFREQUENCY	TIME DEPENDENT	FREQUENCY RESPONSE	PARAMETRIC	QUASI-STATIC TRANSIENT	LINEAR BUCKLING	ELASTO-PLASTIC MATERIAL	FATIGUE	MULTIPHYSICS
Component parametric	48*	2 s	Plane Stress					√					
Component quasi-static	53*	6 s	Plane Stress, Heat Transfer						√				√

* this page number refers to the *Structural Mechanics Module User's Guide*.

** these models require COMSOL Script or MATLAB.

***this model requires the AC/DC Module.

We welcome any questions, comments, or suggestions you might have concerning these models. Contact us at `info@comsol.com`.

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

Acoustic-Structure Interaction

This chapter contains models of the interaction between acoustics and structures, often called *acoustic-structure interaction*.

Vibrations of a Disk Backed by an Air-Filled Cylinder

Introduction

The vibration modes of a thin or thick circular disk are well known, and it is possible to compute the corresponding eigenfrequencies to arbitrary precision from a series solution. The same is true for the acoustic modes of an air-filled cylinder with perfectly rigid walls. A more interesting question to ask is: What happens if the cylinder is sealed in one end not by a rigid wall but by a thin disk? This is the question you address in this model.

Note: This model requires the Acoustics Module and the Structural Mechanics Module.

Model Definition

In COMSOL Multiphysics you can model an air-filled cylinder sealed by a thin disk in one end using at least two different approaches. You describe the pressure in the cavity with a Pressure Acoustics application mode, while the model of the disk can be either a thin shell in 3D, using shell elements, or a 2D plate. The latter approach to modeling this acoustic-structure interaction is possible thanks to nonlocal couplings and COMSOL Multiphysics' ability to model in different numbers of space dimensions at the same time—*extended multiphysics*.

In Ref. 1, D. G. Gorman and others have thoroughly investigated the model at hand, and they have developed a semi-analytical solution verified by experiments and simulations. The geometry is a rigid steel cylinder with a height of 255 mm and a radius of 38 mm. One end is welded to a heavy slab, while the other is sealed with a steel disk only 0.38 mm thick. Some of the theoretical eigenfrequencies of a thin disk in vacuum and of a rigidly sealed chamber are given in the following table (according to Ref. 1).

TABLE 2-1: BENCHMARK VALUES FOR EIGENFREQUENCIES OF THE DISK AND THE CYLINDER

NUMBER	CLAMPED DISK IN VACUUM (HZ)	RIGIDLY SEALED CYLINDER (HZ)
1	671.8	672.5
2	1398	1345
3	2293	2018
4	2615	2645
5	3356	2690
6	4000	4387

Here you model the coupled system using the extended multiphysics approach. This means that you draw the disk in a 2D geometry and model it with Mindlin-theory DRM-plate elements, while you draw the cylinder in a separate 3D geometry. The acoustics in the cylinder is described in terms of the acoustic (differential) pressure. The eigenvalue equation for the pressure is

$$-\Delta p = \frac{\omega^2}{c^2}p$$

where c is the speed of sound and $\omega = 2\pi f$ defines the eigenfrequency, f.

A first step is to calculate the eigenfrequencies for the disk and the cylinder separately and compare them with the theoretical values in Table 2-1. This way you can verify the basic components of the model and assess the accuracy of the FEM solution before modeling the coupled system.

Results and Discussion

Most of the modes show rather weak coupling between the structural bending of the disk and the pressure field in the cylinder. It is, however, interesting to note that some of the uncoupled modes have been split into one co-vibrating and one contra-vibrating mode with distinct eigenfrequencies. This is the case for modes 1 and 2 and for modes 9 and 12 in the FEM solution. The table below shows a comparison of the eigenfrequencies from the COMSOL Multiphysics analysis with the semi-analytical and experimental frequencies reported by D. G. Gorman and others in Ref. 1. The table also states whether the modes are structurally dominated (str), acoustically dominated (ac), or tightly coupled (str/ac).

TABLE 2-2: RESULTS FROM SEMI-ANALYTICAL AND COMSOL MULTIPHYSICS ANALYSES AND EXPERIMENTAL DATA

TYPE	SEMI-ANALYTICAL (HZ)	COMSOL MULTIPHYSICS (HZ)	EXPERIMENTAL (HZ)
str/ac	636.9	637.2	630
str/ac	707.7	707.7	685
ac	1347	1347.4	1348
str	1394	1395.3	1376
ac	2018	2018.6	2040
str	2289	2293.1	2170
str/ac	2607	2612.1	2596
ac	2645	2646.3	–
str/ac	2697	2697.1	2689
ac	2730	2730.9	2756
ac	2968	2969.3	2971

As the table shows, the FEM solution is in good agreement with both the theoretical predictions and the experimentally measured values for the eigenfrequencies. As you might expect from the evaluation of the accuracies for the uncoupled problems, the precision is generally better for the acoustics-dominated modes.

Reference

1. D. G. Gorman, J. M. Reese, J. Horacek, and D. Dedouch: “Vibration analysis of a circular disk backed by a cylindrical cavity,” *Proc. Instn. Mech. Engrs.*, vol. 215, Part C, 2001.

Model Library path: `Structural_Mechanics_Module/Acoustic-Structure_Interaction/coupled_vibration`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **2D** from the **Space dimension** list.

2 In the list of application modes, select **Structural Mechanics Module>Mindlin Plate>Eigenfrequency analysis**.

3 Click **OK**.

GEOMETRY MODELING

The geometry of the disk is a solid circle. Its location does not really matter because you embed it in 3D using coupling variables, but the transformations is trivial if you center the disk at the origin:

1 Press the Shift key and click the **Ellipse/Circle (Centered)** button.

2 In the **Circle** dialog box, type `0.038` in the **Radius** edit field and click **OK** to create a circle of radius 0.038 m, centered at the origin.

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

PHYSICS SETTINGS

Boundary Conditions

The edges of the disk are welded to the cylinder and can therefore be described as rigidly *clamped* or fixed.

1 From the **Physics** menu, choose **Boundary Settings**.

2 Select one of the boundaries and then press Ctrl+A to select all boundaries.

3 Make sure that you have selected **Tangent and normal coord. sys. (t,n)** in the **Coordinate system** list.

4 Select **Fixed** in the **Condition** list.

5 Click **OK**.

Subdomain Settings—Material Properties

The steel disk has the following material properties (in the default SI units):

- Young's modulus, $E = 2.1 \cdot 10^{11}$
- Poisson's ratio, $\nu = 0.3$
- Density, $\rho = 7800$

1 From the **Physics** menu, choose **Subdomain Settings**.

2 Select Subdomain 1.

3 Enter material data according to the following table:

MATERIAL PROPERTY	VALUE
E	`2.1e11`
ν	`0.3`
ρ	`7800`
thickness	`0.00038`

4 Click **OK**.

MESH GENERATION

To obtain accurate values of the eigenfrequencies of the disk, you need a mesh that is finer than the one produced with the default settings.

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

2 Click the **Custom mesh size** option button and type `0.002` in the **Maximum element size** edit field.

3 Click the **Remesh** button and then click **OK**.

COMPUTING THE SOLUTION

When solving for the eigenfrequencies of the disk in vacuum, only the frequency interval between 500 Hz and 3250 Hz is of interest. Start by searching for the 20 first eigenfrequencies (some of these are almost identical and come from double eigenvalues) and make the solver start its search around 500 Hz:

1 From the **Solve** menu, choose **Solver Parameters**.

2 Type `20` in the **Desired number of eigenfrequencies** edit field.

3 Type `500` in the **Search for eigenfrequencies around** edit field.

4 Click **OK**.

5 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 Click the **3D Surface Plot** button to see the deflection of the disk.

2 From the **Postprocessing** menu, choose **Plot Parameters**.

3 Try looking at some of the eigenmodes by selecting the corresponding eigenfrequencies on the **General** page of the **Plot Parameters** dialog box. To do so, open the **Plot Parameters** dialog box from the **Postprocessing** menu. On the **General**

page, choose the eigenfrequencies from the **Eigenfrequency** list in the **Solution to use** area.

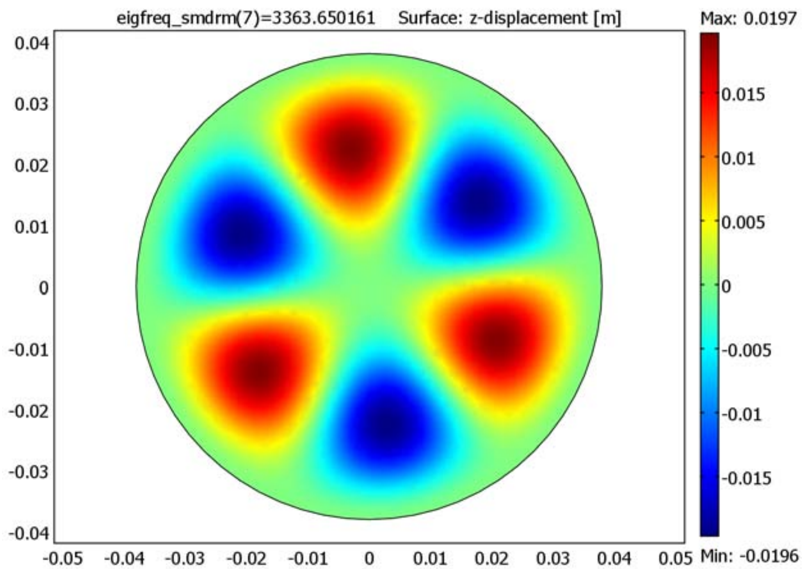


Figure 2-1: The eigenmode associated with the eigenfrequency around 3356 Hz.

You can now compare the eigenfrequencies with the theoretical values for a thin disk. The discrepancy is below 2% for all eigenmodes in the interval, so the conclusion is that the mesh resolution is sufficient.

Adding the 3D Pressure Acoustics Application Mode

Now add a second geometry that will contain the cylinder and the acoustic pressure variable using a 3D Pressure Acoustics application mode.

1 From the **Multiphysics** menu, choose **Model Navigator**.
2 Click the **Add Geometry** button to add a second geometry.
3 In the **Add Geometry** dialog box, select **3D** from the **Space dimension** list.
4 Click **OK**.
5 In the list of application modes, select **Acoustics Module>Pressure Acoustics>Eigenfrequency analysis**.
6 Click **Add**.

7 Click **OK**.

GEOMETRY MODELING

1 Click the **Cylinder** toolbar button.

2 Type `0.038` in the **Radius** edit field and `0.255` in the **Height** edit field.

3 Click **OK**.

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

PHYSICS SETTINGS

Boundary Conditions

For the moment, assume that all boundaries are perfect hard walls, which is the default boundary condition.

Subdomain Settings

1 From the **Physics** menu, choose **Subdomain Settings**.

2 Select Subdomain 1.

3 Type `1.2` in the **Fluid density** (ρ_0) edit field. Leave the other properties at their default value.

4 Click **OK**.

MESH GENERATION

Click the **Initialize Mesh** button to create a mesh using the default parameters.

COMPUTING THE SOLUTION

To solve for the acoustic modes only, you must deactivate the Mindlin Plate application mode during the solution. If the plate is not deactivated, COMSOL Multiphysics solves the two eigenvalue problems simultaneously but independently of one another.

1 From the **Solve** menu, choose **Solver Manager**.

2 Click the **Solve For** tab.

3 Ctrl-click on the **Mindlin Plate (smdrm)** folder to clear that application mode's variables from the list of variables to solve for, then click **OK**.

4 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

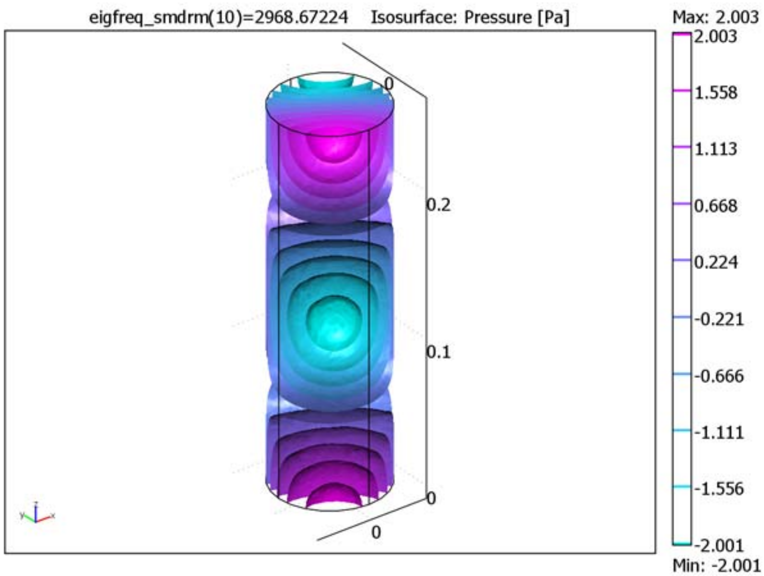
1 Clear the **Slice** check box and select the **Isosurface** check box on the **General** page of the **Plot Parameters** dialog box.

2 In the **Solution to use** area select one of the solutions near 2730 Hz from the **Eigenfrequency** list.

3 Click the **Isosurface** tab, and type `10` in the edit field for isosurface levels under **Number of levels**. From the **Colormap** list select **cool**, then click **Apply**.

4 Click the **Headlight** button on the Camera toolbar.

5 Try some of the different eigenfrequencies by selecting the corresponding eigenfrequencies on the **General** page of the **Plot Parameters** dialog box.



You can also compare these eigenfrequencies with the theoretical values in Table 2-2 on page 12. This time, the relative error seems to be much smaller than for the disk, which means that any additional mesh refinement should be done on the plate part.

Coupling the Equations

The first step in the process of coupling the Mindlin plate elements to the acoustic equation is to create the nonlocal couplings; use coupling variables to make the pressure available as a load on the plate and the out-of-plane displacement of the plate a valid parameter in the coefficients for the acoustic equation.

OPTIONS AND SETTINGS—COUPLING VARIABLES

First define a coupling variable for the acoustic pressure from the top face of the cylinder to the disk (Mindlin plate):

1. On the **Options** menu, point to **Extrusion Coupling Variables** and then click **Boundary Variables**.
2. In the **Boundary Extrusion Variables** dialog box, select Boundary 4 (the top face) of the cylinder.
3. Type `p` in the top row under both **Name** and **Expression**.
4. Click the **Destination** tab.
5. Select **Geom1** in the **Geometry** list, then select Subdomain 1 (the disk) in the 2D geometry.
6. Click the **Source Vertices** tab.
7. In the **Vertex selection** list, select Vertices 2, 4, and 8. Click the **>>** button.
8. Click the **Destination Vertices** tab.
9. In the **Vertex selection** list, select Vertices 1, 2, and 4. Click the **>>** button.
10. Click **OK**.

Now define a coupling variable for the out-of-plane displacement, w, from the disk to the top face of the cylinder. Also the acceleration, w_{tt}, is needed.

1. If you are in the Pressure Acoustics application mode, switch to the Mindlin Plate application mode by choosing **Geom1: Mindlin Plate (smdrm)** from the **Multiphysics** menu.
2. On the **Options** menu, point to **Extrusion Coupling Variables** and then click **Subdomain Variables**.
3. In the **Subdomain Extrusion Variables** dialog box, select Subdomain 1.
4. Type `w` in the top row under both **Name** and **Expression**. Type `wtt` in the second row under both **Name** and **Expression**.
5. Click the **Destination** tab.
6. Select `w` from the **Variable** list.
7. Select **Geom2** in the **Geometry** list, then select the check box next to Boundary 4.
8. Click the **Source Vertices** tab.
9. In the **Vertex selection** list, select Vertices 1, 2, and 4. Click the **>>** button.
10. Click the **Destination Vertices** tab.
11. In the **Vertex selection** list, select Vertices 2, 4, and 8. Click the **>>** button.

12 Click the **Destination** tab.

13 Select `wtt` from the **Variable** list.

14 Select **Geom2** in the **Geometry** list, then select he check box next to Boundary 4.

15 Click the **Source Vertices** tab.

16 In the **Vertex selection** list, select Vertices 1, 2, and 4. Click the **>>** button.

17 Click the **Destination Vertices** tab.

18 In the **Vertex selection** list, select Vertices 2, 4, and 8. Click the **>>** button.

19 Click **OK**.

PHYSICS SETTINGS

Boundary Conditions

The sound-hard boundary condition for a rigid wall is that the normal acceleration vanishes. For a moving wall, such as the thin disk that now seals the cylinder, the appropriate condition is instead

$$\frac{\mathbf{n} \cdot \nabla p}{\rho_a} = -a$$

where a is the normal acceleration of the wall.

1 If you are in the Mindlin Plate application mode, switch to the Pressure Acoustics application mode by choosing **Geom2: Pressure Acoustics (acpr)** from the **Multiphysics** menu.

2 On the **Physics** menu, point to **Boundary Settings**.

3 Select the top of the cylinder where the plate is located, that is, Boundary 4.

4 Select **Normal acceleration** from the **Boundary condition** list.

5 Type `-wtt` (the structural acceleration in the negative z direction) in the **Inward acceleration** (a_n) field and click **OK**.

Subdomain Settings

The coupling in the other direction is described by the acoustic pressure acting as a normal load.

1 Choose the **Geom1: Mindlin Plate (smdrm)** application mode from the **Multiphysics** menu.

2 Open the **Subdomain Settings** dialog box.

3 Click the **Load** tab.

4 Select Subdomain 1.

5 Type `p` in the **Fz** edit field as a surface load on the disk.

6 Click **OK**.

7 Switch back to the 3D geometry by choosing **Geom2: Pressure Acoustics (acpr)** from the **Multiphysics** menu.

COMPUTING THE SOLUTION

1 From the **Solve** menu, choose **Solver Manager**.

2 In the **Solver Manager** dialog box, click the **Solve For** tab.

3 Reactivate the Mindlin Plate application mode by selecting both **Geom1(2D)** and **Geom2(3D)** and all corresponding variables in the **Solve for** list.

4 Click the **Solve** button to compute the solution. When done, click **OK**.

POSTPROCESSING AND VISUALIZATION

1 Open the **Plot Parameters** dialog box.

2 Add boundary and deformed shape plots in addition to the isosurface plot by selecting the **Boundary** and **Deformed shape** check boxes.

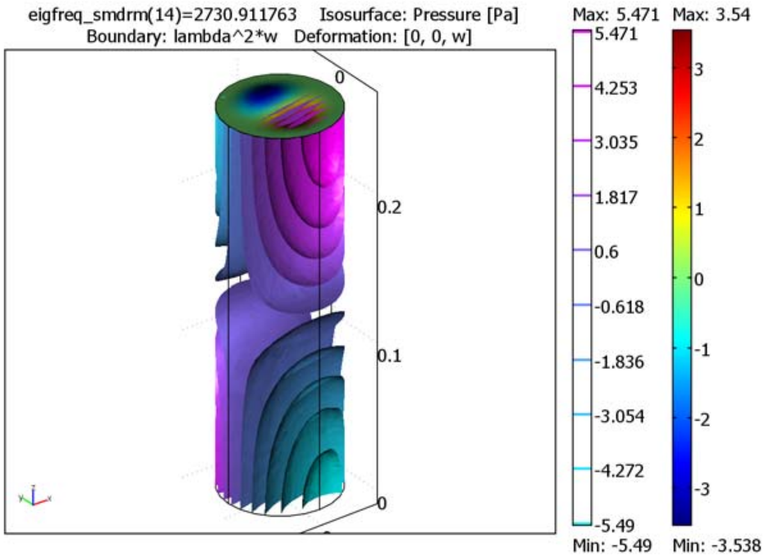
3 Click the **Boundary** tab.

4 Type `lambda^2*w` in the **Expression** edit field in the **Boundary data** area, that is, the normal acceleration of the disk. On the other boundaries w is not defined so those boundaries are invisible.

5 Click the **Deform** tab and select the **Boundary** check box only in the **Domain types to deform** area.

6 In the **Deformation data** area, click the **Boundary Data** tab and type `0`, `0`, and `w` in the **x component**, **y component**, and **z component** edit field, respectively.

7 To examine the different eigenmodes, click the **General** tab and select an eigenfrequency from the **Eigenfrequency** list. Click **Apply** to plot the solution.



Acoustic-Structure Interaction

Introduction

Liquid or gas acoustics coupled to structural objects such as membranes, plates, or solids represents an important application area in many engineering fields. Some examples include:

- Loudspeakers
- Acoustic sensors
- Nondestructive impedance testing
- Medical ultrasound diagnostics of the human body

Model Definition

This model provides a general demonstration of an acoustic fluid phenomenon in 3D that is coupled to a solid object. The object's walls are impacted by the acoustic pressure. The model calculates the frequency response from the solid and then feeds this information back to the acoustics domain so that it can analyze the wave pattern. As such, the model becomes a good example of a scattering problem.

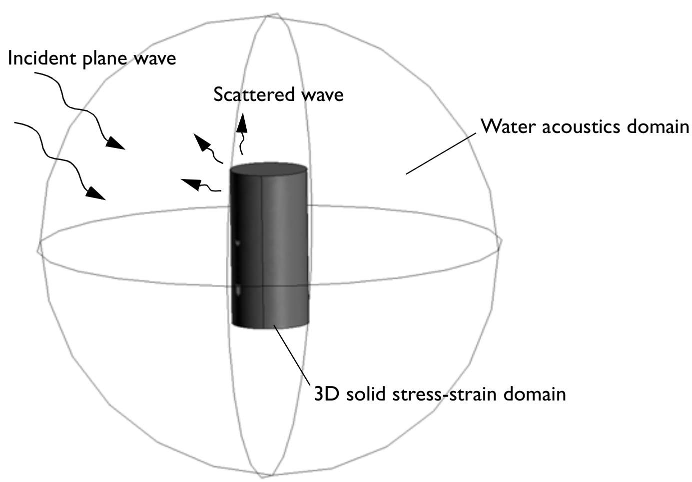


Figure 2-2: Geometric setup of an aluminum cylinder immersed in water.

Figure 2-2 illustrates an aluminum cylinder immersed in water. The incident wave is 60 kHz, in the ultrasound region. The cylinder is 2 cm high and has a diameter of 1 cm. The water acoustic domain is truncated as a sphere with a reasonably large diameter. What drives the system is an incident plane wave from the surroundings into the spherical boundary. The harmonic acoustic pressure in the water on the surface of the cylinder acts as a boundary load in the 3D solid to ensure continuity in pressure. The model calculates harmonic displacements and stresses in the solid cylinder, and it then uses the normal acceleration of the solid surface in the acoustics domain boundary to ensure continuity in acceleration.

DOMAIN EQUATIONS

Water Subdomain

For harmonic sound waves we use the frequency-domain Helmholtz equation for sound pressure

$$\nabla \cdot \left(-\frac{1}{\rho_0}\nabla p + \mathbf{q}\right) - \frac{\omega^2 p}{\rho_0 c^2} = 0$$

where the acoustic pressure is a harmonic quantity, $p = p_0 e^{i\omega t}$, and p is the pressure (N/m^2), ρ_0 is the density (kg/m^3), $\mathbf{q}$ is an optional *dipole source* (m/s^2), ω is the angular frequency (rad/s), and c is the speed of sound (m/s).

TABLE 2-3: ACOUSTICS DOMAIN DATA

QUANTITY	VALUE	DESCRIPTION
ρ_0	997 kg/m^3	Density
c	1500 m/s	Speed of sound
$f=\omega/2\pi$	60 kHz	Frequency

Solid Subdomain

In the solid cylinder you calculate the harmonic stresses and strains using a frequency response analysis in the 3D Solid, Stress-Strain application mode. The material data comes from the built-in database for Aluminum 3003-H18.

BOUNDARY CONDITIONS

Outer Perimeter

On the outer spherical perimeter of the water domain (Figure 2-2) we specify an incident plane wave to represent an incoming sound wave. A superimposed spherical wave is allowed to travel out of the system as a response from the cylinder. In

COMSOL Multiphysics' Acoustics application mode you implement this scenario by using the prepared *Radiation condition* with the *Spherical wave* option. The radiation boundary condition is useful when the surroundings are only a continuation of the domain.

TABLE 2-4: RADIATION BOUNDARY CONDITION SETTINGS

QUANTITY	VALUE	DESCRIPTION
$\hat{k}$	$(\sin\theta\cos\varphi, \sin\theta\sin\varphi, \cos\theta)$	Incident wave direction vector
p_0	1 Pa	Pressure amplitude

The incident wave direction is controlled by the two angles $0 < \theta < 2\pi$ and $0 < \varphi < \pi$.

For mathematical details on the radiation boundary condition, see the section "Radiation Boundary Conditions" on page 26 of the *COMSOL Multiphysics Modeling Guide*.

Interface Cylinder-Water

To couple the sound-pressure wave to the solid cylinder, set the boundary load $\mathbf{F}$ (force/unit area) on the solid cylinder to

$$\mathbf{F} = -\mathbf{n}_s p$$

where $\mathbf{n}_s$ is the outward-pointing unit normal vector seen from inside the solid domain.

To couple back the frequency response of the solid to the acoustics problem, use a normal acceleration boundary condition

$$-\mathbf{n}_a \cdot \left(-\frac{1}{\rho_0}\nabla p + \mathbf{q}\right) = a_n$$

where $\mathbf{n}_a$ is the outward-pointing unit normal vector seen from inside the acoustics domain. Also set the normal acceleration a_n to $(\mathbf{n}_a \cdot \mathbf{u})\omega^2$, where $\mathbf{u}$ is the calculated harmonic-displacement vector of the solid structure.

HARD-WALL COMPARISON

As a reference we also study a simpler model where the solid interface is regarded as a hard wall. In this model we turn off the structural analysis of the cylinder, and we set the cylinder surface to an acoustic hard wall with the boundary condition

$$\mathbf{n}_a \cdot \left(-\frac{1}{\rho_0}\nabla p + \mathbf{q}\right) = 0\,.$$

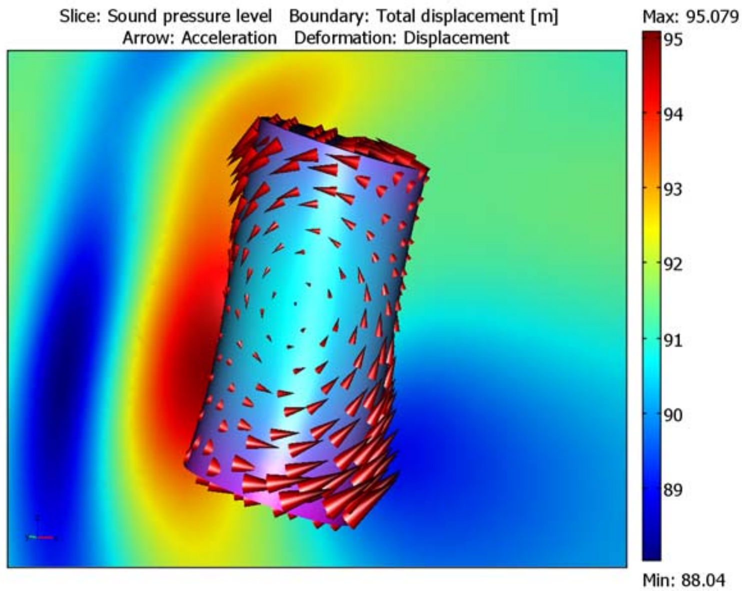


Figure 2-3: Sound-pressure plot (dB) of the acoustic waves in the coupled problem. The cone lengths are proportional to the surface acceleration, which is a direct measure of the sound-pressure interaction between the water and the cylinder.

Figure 2-3 displays the sound pressure as a slice plot. It is clear from which direction the sound wave propagates into the domain. The values of the deformation are very small, but the acceleration is enough to have an impact on the sound waves.

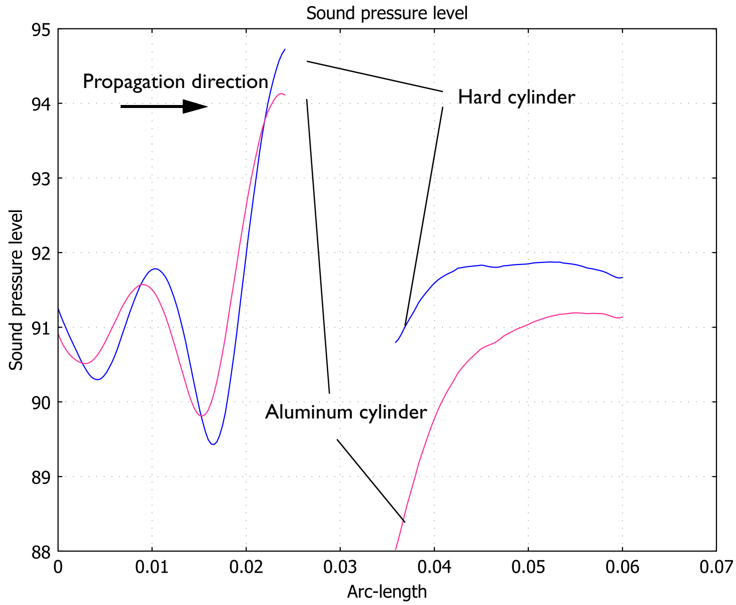


Figure 2-4: Sound pressure level on impact and on the shadow side of the cylinder.

Figure 2-4 shows a comparison between the hard-wall example and the full aluminum solid model. Near the cylinder wall the plot shows that the sound pressure level is higher on the upstream side for the hard-wall case than for the aluminum model. Inversely, the amplitude is lower for the hard-wall model than for the aluminum model on the downstream side. This shows that the hard wall reflects more and transmits less energy than the aluminum cylinder. The conclusion is that the mechanical properties of the metal object have an impact on the acoustic signature.

Modeling in COMSOL Multiphysics

COMSOL Multiphysics provides specialized direct solvers for symmetric systems. You can employ such solvers for problems that generate symmetric stiffness matrices and thereby save a considerable amount of system memory and shorten the calculation time.

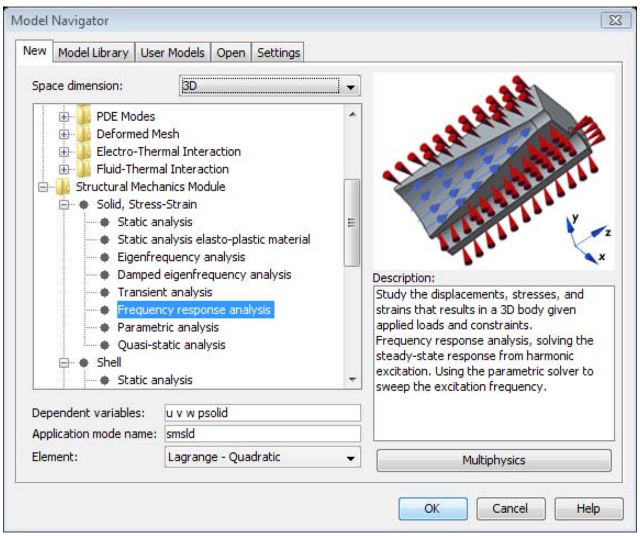
Model Library path: `Structural_Mechanics_Module/Acoustic-Structure_Interaction/acoustic_structure`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Start COMSOL Multiphysics.

2 In the **Model Navigator**, select **3D** from the **Space dimension** list, then select **Structural Mechanics Module>Solid, Stress-Strain>Frequency response analysis** in the list of application modes.



3 Type `u v w psolid` in the **Dependent variables** edit field.

4 Click the **Multiphysics** button, then click **Add**.

5 Select **COMSOL Multiphysics>Acoustics>Acoustics** in the list of application modes, click the **Add** button, then click **OK**.

OPTIONS AND SETTINGS

1 Open the **Constants** dialog box from the **Options** menu and enter the following values (the descriptions are optional):

NAME	EXPRESSION	DESCRIPTION
Freq	60[kHz]	Frequency
phi	(-pi/6)[rad]	Wave direction angle, phi
theta	(4*pi/6)[rad]	Wave direction angle, theta
rhow	997[kg/m^3]	Density
cw	1500[m/s]	Speed of sound
k1	sin(theta)*cos(phi)	Incident wave direction vector, x component
k2	sin(theta)*sin(phi)	Incident wave direction vector, y component
k3	cos(theta)	Incident wave direction vector, z component

2 Click **OK**.

3 Choose the **Physics>Scalar Variables** menu item. Enter the items from the following table.

NAME	EXPRESSION	DESCRIPTION	UNIT
freq_smsld	Freq	Excitation frequency	Hz
freq_aco	Freq	Excitation frequency	Hz
p_ref_aco	20e-6	Pressure reference	Pa

4 Click **OK**.

GEOMETRY MODELING

1 Click the **Cylinder** button on the left toolbar. In the dialog box that appears specify the following values:

Radius	0.005
Height	0.02
Axis base point, z	-0.01

and let all other entries retain their default values. Click **OK**, then click the **Zoom Extents** button on the main toolbar.

2 Click the **Sphere** button on the left toolbar. In the dialog box that appears specify a **Radius** of `0.03` and let the other entries retain their default values. Click **OK**, then click the **Zoom Extents** button on the Main toolbar.

PHYSICS SETTINGS—HARD-WALL CASE

Subdomain Settings

1 Select the **Multiphysics** menu and **Solid, Stress-Strain (smsld)**.

2 Select the **Subdomain Settings** on the **Physics** menu. Select all subdomains by selecting one subdomain, and then pressing Ctrl+A. Clear the **Active in this domain** check box, then click **OK**.

3 Select the **Multiphysics** item menu and then select **Acoustics (aco)**.

4 Select the **Physics>Subdomain Settings** menu, then select Subdomain 2, and finally clear the **Active in this domain** check box.

5 Select Subdomain 1 and enter the following data:

QUANTITY	VALUE/EXPRESSION
ρ_0	rhow
c_s	cw
q	0 0 0

6 Click **OK**.

Boundary Conditions

1 Select the menu item **Physics>Boundary Settings.** Hold down the Ctrl key and select Boundaries 1–4, 9, 10, 12, and 13. Select the boundary condition **Radiation condition** with **Wave type: Spherical wave**. Fill out the dialog box with values from the following table:

QUANTITY	VALUE/EXPRESSION
p_0	1
x_0	0
y_0	0
z_0	0
n_k	k1 k2 k3

2 Leave the remainder of the boundaries at their default value **Sound hard boundary (wall)**.

3 Click **OK**.

GENERATING THE MESH

1 Select the **Mesh>Free Mesh Parameters** menu item. On the **Global** page select **Coarse** from the **Predefined mesh sizes** list.

2 Go to the **Subdomain** page and set the **Maximum element size** to `0.005` for Subdomain 1 and to `0.003` for Subdomain 2. Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button.

2 Select **Stationary** in the **Solver** list.

3 Click **OK**.

4 Click the **Solve** button on the Main toolbar.

Postprocessing

To render the hard-wall line in Figure 2-4 on page 26 follow these steps:

1 Select the menu item **Postprocessing>Cross-Section Plot Parameters**.

2 On the **General** page click the **Line/Extrusion plot** option button, then select the **Keep current plot** check box.

3 Go to the **Line/Extrusion** page and click the **Line plot** option button.

4 From the **Predefined quantities** list select **Acoustics (aco)>Sound pressure level**.

5 Define the line through the origin that coincides with the incident-wave propagation vector $\hat{k}$. You can conveniently take the vector components from the **Value** column for the constants k_1, k_2, and k_3 in the **Constants** list in the **Options** menu.

x0, x1	`0.03*(-.75)`	`0.03*.75`
y0, y1	`0.03*.433013`	`0.03*(-.433013)`
z0, z1	`0.03*.5`	`0.03*(-.5)`

6 Click **OK**.

7 Leave this figure window open in the background during the next stage of the solution.

PHYSICS SETTINGS—COUPLED ACOUSTICS-SOLID

Subdomain Settings

1 Select the **Multiphysics** menu, then select **Solid, Stress-Strain (smsld)**.

2 Choose **Physics>Subdomain Settings**. Select Subdomain 2, then select the **Active in this domain** check box.

3 Click **Load** and select **Aluminum 3003-H18** from the **Basic Material Properties** list. Click **OK**.

4 Click the **Damping** tab. Select **No damping** from the **Damping model** list.

5 Click **OK**.

Boundary Conditions

1 Select the menu item **Physics>Boundary Settings**. Select Boundaries 5–8, 11, and 14, then go to the **Load** page. Specify **F_x:** `-p*nx_smsld`, **F_y:** `-p*ny_smsld`, and **F_z:** `-p*nz_smsld`, then click **OK**.

   `p` is the dependent variable for the pressure, and `nx_smsld`, `ny_smsld`, and `nz_smsld` represent the outward unit normal pointing out from the structural domain.

2 Select the **Multiphysics** menu and then select **Acoustics (aco)**.

3 Select the menu item **Physics>Boundary Settings**. Select the **Select by group** check box. Select Boundary 5 to get a group selection of all the hard-wall boundaries.

4 Set the boundary condition to **Normal acceleration**, then specify a_n as `nx_smsld*u_tt_smsld+ny_smsld*v_tt_smsld+nz_smsld*w_tt_smsld`. Click **OK**.

   Here, `u_tt_smsld`, `v_tt_smsld`, and `w_tt_smsld` are the structural acceleration components in the x, y, and z directions, respectively, so the expression that you enter corresponds to $a_n = \mathbf{n} \cdot \mathbf{u}_{tt}$.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to start the analysis.

Postprocessing

To overlay the sound pressure of the coupled problem on the hard-wall problem in Figure 2-4 on page 26 follow these steps:

1 Choose **Postprocessing>Cross-Section Plot Parameters**.

2 On the **General** page click the **Line/Extrusion plot** option button. Click **OK**.

To generate Figure 2-3 on page 25, do as follows:

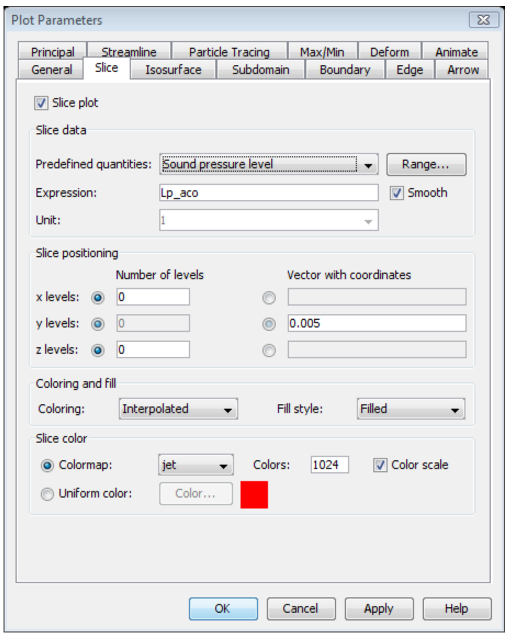
1 Choose **Options>Suppress>Suppress Boundaries**.

2 Select Boundaries 1, 2, 9, and 10 from the list. Click **OK**.

3 Click the **Plot Parameters** button. On the **General** page, select the check boxes for **Slice**, **Boundary**, **Arrow**, and **Deformed shape.** Leave all the other check boxes unchecked.

4 Clear the **Element refinement: Auto** check box, then type `7` in the **Refinement** edit field.

5 Click the **Slice** page, then select **Acoustics (aco)>Sound pressure level** from the **Predefined quantities** list. In the **Slice** positioning area type `0` in all three **Number of levels** edit fields. On the **y-levels** line click **Vector with coordinates** and type `0.005` in the **y-levels** position.



6 Go to the **Boundary** page and from the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>Total displacement**. Use **Colormap: cool**.

7 Go to the **Arrow** page.

8 Select **Boundaries** from the **Plot arrows on** list.

9 Click the **Boundary Data** tab, then select **Acceleration** from the **Predefined quantities** list.

10 In the **Arrow parameters** area, select **Cone** from the **Arrow type** list and set the **Arrow length** to **Proportional**. Clear the **Scale factor: Auto** check box, then enter `0.6` as the **scale factor**.

11 Click the **Deform** tab.

12 Go to the **Domain types to deform** area and make sure that only the **Boundary** check box is selected.

13 Clear the **Scale factor: Auto** check box, then type `3.8e9` in the associated edit field.

14 Click **OK**.

To refine the image's visual quality do as follows:

1 Click both the **Headlight** and **Scene Light** buttons on the Camera toolbar.

2 Select the menu item **Options>Visualization/Selection Settings**.

3 Go to the **Camera** page and select **Projection: Perspective**.

4 Go to the **Lighting** page. In the **Scene light** area, click all four light sources and clear the **Enabled** check box on each of them.

5 Click the **New** button. Select **Type: Spot**, then click **OK**.

6 Specify the light source as in the following table, then click **OK**.

PROPERTY	VALUE
Position	`-0.01 -0.01 0`
Direction	`0 1 0`
Spread angle	90
Concentration	`0.05`

You can experiment with the view angle by clicking the **Zoom** and **Dolly In/Out** buttons on the Plot toolbar and clicking and dragging the geometry.

Piezoacoustic Transducer

Introduction

A piezoelectric transducer can be used either to transform an electric current to an acoustic pressure field or, the opposite, to produce an electric current from an acoustic field. These devices are generally useful for applications that require the generation of sound in air and liquids. Examples of such applications include phased array microphones, ultrasound equipment, inkjet droplet actuators, drug discovery, sonar transducers, bioimaging, and acousto-biotherapeutics.

Model Definition

In a phased-array microphone, the piezoelectric crystal plate fits into the structure through a series of stacked layers that are divided into rows. The space between these layers is referred to as the *kerf*, and the rows are repeated with a periodicity, or *pitch*.

This model simulates a single crystal plate in such a structure. The element is rotationally symmetric, making it possible to use an axisymmetric 2D application mode in COMSOL Multiphysics.

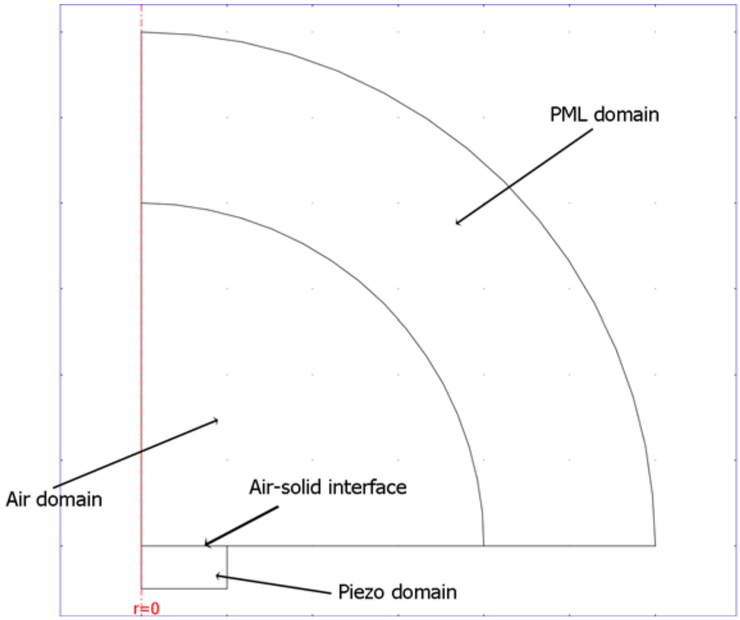


Figure 2-5: The model geometry.

In the air domain, the wave equation describes the pressure distribution:

$$\frac{1}{\rho_0 c_s^2}\frac{\partial^2 p}{\partial t^2} + \nabla\cdot\left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) = Q \tag{2-1}$$

For this model, assume that the pressure varies harmonically in time as

$$p(\mathbf{x}, t) = p(\mathbf{x})e^{i\omega t}$$

Hence equation 2-1 simplifies to

$$\nabla\cdot\left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) - \frac{\omega^2 p}{\rho_0 c_s^2} = Q \tag{2-2}$$

Because there are no sources present, Equation 2-2 simplifies further to

$$\nabla\cdot\left(-\frac{1}{\rho_0}(\nabla p)\right) - \frac{\omega^2 p}{\rho_0 c_s^2} = 0$$

The piezoelectric domain is made of the crystal PZT5-H, which is a common material in piezoelectric transducers. The structural analysis is also time harmonic although, for historical reasons, in structural-mechanics terminology it is a frequency response analysis.

The frequency is set to 300 kHz, which is in the ultrasonic range (dolphins and bats, for example, communicate in the range of 20 Hz to 150 kHz, while humans can only hear frequencies in the range 20 Hz to 20 kHz).

BOUNDARY CONDITIONS

A voltage of 100 V is applied to the upper part of the transducer, while the bottom part is grounded. At the interface between the air and solid domain, the boundary condition for the acoustics application mode is that the pressure is equal to the normal acceleration of the solid domain

$$n \cdot \left(\frac{1}{\rho_0}(\nabla p)\right) = a_{\mathrm{n}}$$

where a_{n} is the normal acceleration.

This drives the pressure in the air domain. The solid domain is on the other hand subjected to the acoustic pressure changes in the air domain. Because of the high voltage applied to the transducer, this load is probably negligible in comparison. Yet because the model is in 2D, it is possible to include this load solve the full model simultaneously on any computer.

Results and Discussion

Figure 2-6 shows the pressure distribution in the air domain. This plot clearly shows how the PML (perfectly matched layer) absorbs the wave effectively.

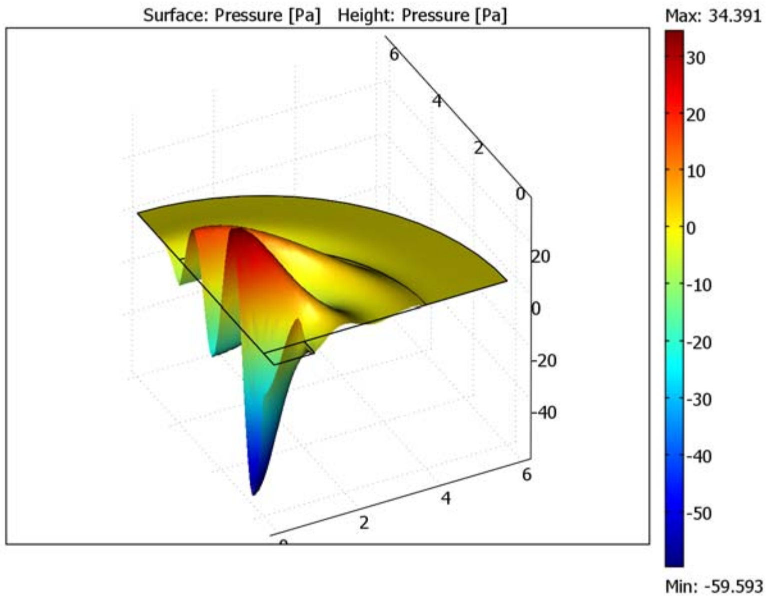


Figure 2-6: Surface and height plot of the pressure distribution.

Figure 2-7 shows the pressure distribution along the air-solid interface. The acoustic pressure load is small in comparison to the electrical load, which is plotted in Figure 2-8 on page 38.

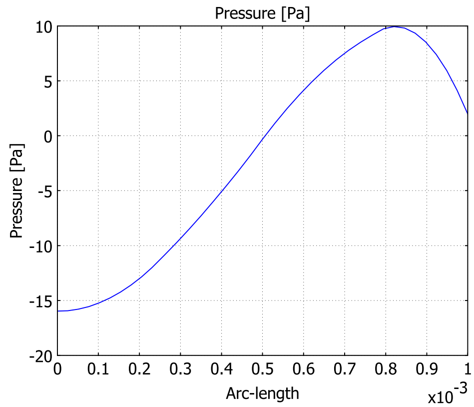


Figure 2-7: Acoustic pressure at the air-solid interface.

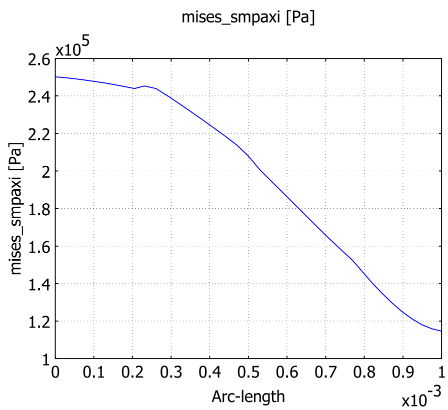


Figure 2-8: von Mises Stress along the air-solid interface.

The results from a far-field analysis appear in Figure 2-9 on page 39. This figure shows that the sound pressure level reaches a maximum right in front of the transducer. This result also shows that the sound pressure level is fairly low. Although humans cannot here this high frequencies, it can be mentioned for comparison that 15 dB is about the same sound pressure level as rustling leaves. On the other hand, because this is just one element in an array of elements, a more detailed study is necessary in order to draw further conclusions.

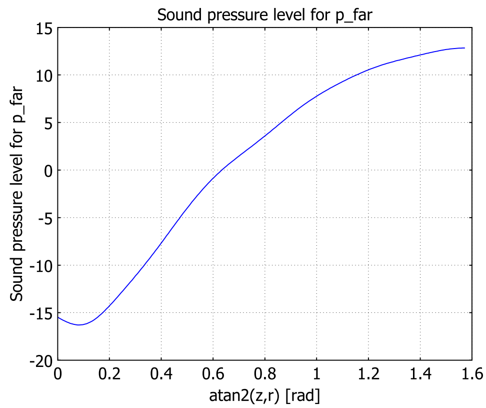


Figure 2-9: The far-field sound pressure level.

Model Library path: `Structural Mechanics Module/Acoustic-Structure Interaction/piezoacoustic_transducer`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. In the **Model Navigator**, begin by selecting **Axial symmetry (2D)** from the **Space dimension** list, then click the **Multiphysics** button.
2. Navigate to **Structural Mechanics Module>Piezoelectric Effects>Piezo Axial Symmetry>Frequency Response Analysis**, then click **Add**.
3. Select **Acoustics Module>Pressure Acoustics>Time-harmonic analysis**; then click **Add**.
4. Click **OK** to close the **Model Navigator**.

SCALAR VARIABLES

1. From the **Physics** menu, select **Scalar Variables**.
2. Select the **Synchronize equivalent variables** check box.
3. Enter a value of `200e3` Hz for the excitation frequency **freq_smpaxi**.

   COMSOL Multiphysics automatically updates the other excitation frequency, **freq_acpr**, to the same value.

4 Click **OK**.

GEOMETRY MODELING

1 Draw a rectangle, R1, by first selecting **Draw>Specify Objects>Rectangle** and then specifying the following properties; when done, click **OK**.

PROPERTY	VALUE
Width	`1e-3`
Height	`0.5e-3`
Position, base	Corner
Position, r	`0`
Position, z	`-0.5e-3`

2 Click the **Zoom Extents** button on the Main toolbar to automatically fit the geometry to your window.

The geometry for the transducer is now complete. Continue by creating the acoustics domain, which consists of two domains: one air domain and one PML domain.

3 Choose **Draw>Specify Objects>Circle**. Specify a **Radius** of `4e-3`, then click **OK**.

4 Choose **Draw>Specify Objects>Square**. Specify a **Width** of `4e-3`, then click **OK**.

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

6 Select the circle and the square, then click the **Intersection** button on the Draw toolbar.

7 Select the geometry object CO1. Press Ctrl+C to copy it, then paste the copy at the same location by pressing Ctrl+V.

8 Select the geometry object CO2, then click the **Scale** button. In the **Scale factor** area, type `1.5` in both the **r** and the **z** edit field. (You can select the geometry objects from a list in the **Create Composite Objects** dialog box.)

9 Once again, click the **Zoom Extents** button on the Main toolbar to automatically fit the geometry to your window.

SUBDOMAIN SETTINGS—PIEZO AXIAL SYMMETRY

1 Select the **Piezo Axial Symmetry (smpaxi)** application mode from the **Model Tree** or from the **Multiphysics** menu.

2 From the **Physics** menu, choose **Subdomain Settings**.

3 Select Subdomains 2 and 3, then clear the **Active in this domain** check box.

4 Select Subdomain 1, then click the **Load** button.

5 From the **Basic Material Properties** library, select **Lead Zirconate Titanate (PZT-5H)**.

In the **Piezoelectric Material Properties** library, you find more than 20 additional piezoelectric materials.

6 Click **OK** to close the **Materials/Coefficients Library** dialog box.

Note: For a piezoelectric material, you can specify an orientation and a coordinate system. In this model, use the default settings: the *xz*-plane in the global coordinate system.

---

7 Click **OK** to close the **Subdomain Settings** dialog box.

BOUNDARY CONDITIONS—PIEZO AXIAL SYMMETRY

1 From the **Physics** menu, choose **Boundary Settings**.

2 On the **Constraint** page, enter the following structural boundary conditions:

SETTINGS	BOUNDARY 1	BOUNDARY 2
Condition	Symmetry Plane	Roller

3 Click the **Electric BC** tab, and set the electric boundary condition as follows:.

SETTINGS	BOUNDARY 1	BOUNDARY 2	BOUNDARY 4	BOUNDARY 6
Type	Axial symmetry	Ground	Electric potential	Zero charge/Symmetry
V0			100	

4 Select Boundary 4.

5 On the **Load** page, type `-p` in the $\mathbf{F_z}$ edit field to specify the acoustic pressure load. `p` is the name of the dependent variable for pressure in the Pressure Acoustics application mode. The pressure acts from the air toward the piezo domain (in the negative *z* direction), which explains the minus sign in front of `p`.

SUBDOMAIN SETTINGS—PRESSURE ACOUSTICS

1 Select the **Pressure Acoustics (acpr)** application mode from the **Model Tree** or from the **Multiphysics** menu.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select Subdomain 1, then clear the **Active in this domain** check box.

4 Select Subdomain 3.

5 On the **PML** page, select **Spherical** from the **Type of PML** list.

6 Select the **Absorbing in radial dir.** check box, then enter a value of `2e-3`. This value corresponds to the PML's extension in the radial direction.

7 Set the inner PML radius, $\mathbf{R_0}$, to `4e-3`.

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

Because the default values correspond to the properties of air, you do not have to specify the subdomain settings for Subdomain 2.

BOUNDARY CONDITIONS—PRESSURE ACOUSTICS

1 From the **Physics** menu, choose **Boundary Settings**.

2 Select Boundary 4, then select **Normal acceleration** as the boundary condition.

3 Set the value of the inward acceleration, $\mathbf{a_n}$, to `w_tt_smpaxi` (this is the second-order time derivative of the structural displacement).

4 Click **OK**.

MESH GENERATION

1 From the **Mesh** menu, choose **Free Mesh Parameters**.

2 On the **Global** page, click the **Custom mesh size** option button.

3 Specify a **Maximum element size** of `(343/200e3)/5`. This value corresponds to 1/5th of the acoustic wavelength. For wave models it is important to use a mesh size sufficiently small to properly resolve the wavelength.

4 Click **Remesh**. When the mesher has finished, click **OK** to close the dialog box.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 From the **Solver** list, select **Stationary**.

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

4 Click the **Solve** button from the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To create figure 2-6, proceed as follows:

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

2 Click the **Surface** tab.

3 On the **Surface Data** page, select **Pressure Acoustics (acpr)>Pressure** from the **Predefined quantities** list.

4 On the **Height Data** page, select the **Height data** check box.
5 From the **Predefined quantities** list, select **Pressure Acoustics (acpr)>Pressure**.
6 Click **OK** to generate the plot.
7 Click the **Headlight** button on the Camera toolbar for clearer visualization.

To create Figure 2-7 and Figure 2-8 do the following:

1 Open the **Domain Plot Parameters** dialog box from the **Postprocessing** menu.
2 Go to the **Line/Extrusion** page.
3 In the **Expression** edit field, enter `p`.
4 From the **Boundary selection** list, select boundary 4 and then click on **Apply**.
5 To plot von Mises stress along the same boundary (Figure 2-8), enter `mises_smpaxi` in the **Expression** edit field, and then click **OK**.

To create Figure 2-9, you need to define a far-field variable.

1 Assure that the **Pressure Acoustics (acpr)** is selected from the **Model Tree**, then open the **Boundary Settings** dialog box.
2 Select Boundary 10 from the list and then click the **Far-Field** tab.
3 Enter `p_far` in the **Name** edit field and then click **OK**.
4 From the **Solve** menu, choose **Update Model**.
5 Open the **Domain Plot Parameters** dialog box from the **Postprocessing** menu.
6 Go to the **Line/Extrusion** page.
7 Select Boundary 10 from the list of boundaries.
8 Select **Sound pressure level for p_far** from the **Predefined quantities** list.
9 Click the **Expression** button in the **x-axis data** area, and then click on the **Expression** button.
10 Enter `atan2(z,r)` in the **Expression** edit field and click **OK**.
11 Click **OK** to create Figure 2-9.

3

Automotive Application Models

This chapter presents models of automotive applications.

Diesel Engine Piston

Introduction

A diesel engine piston is studied at steady-state conditions, that is, at a continuous engine speed and load. The combustion process at steady state produces cyclic pressure loads and a high constant temperature. These load conditions could yield a piston failure due to fatigue cracking, so-called high cycle fatigue cracks.

Model Definition

To reduce the size of this model you can use the symmetries. The model has two symmetry planes, and the geometry is therefore reduced to a quarter of the original geometry.

MECHANICAL LOADS

The applied mechanical loads consist of the following two parts:

- The peak combustion pressure which is applied on the combustion bowl, crown, and top land area (see Figure 3-1).

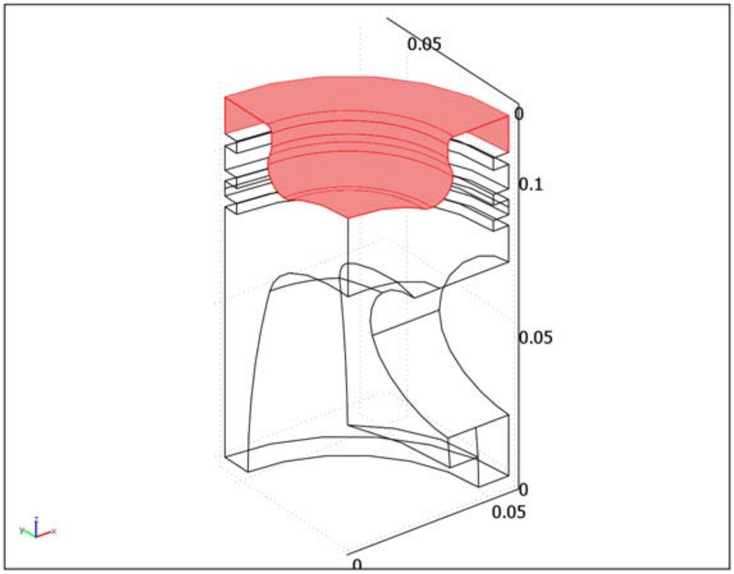


Figure 3-1: Boundaries which are subjected to the combustion pressure.

- The inertia (maximum) load at top dead center, TDC, that is, at the top of the stroke. The acceleration at TDC is calculated from:

$$a = r\omega^2(1 + \lambda)$$

  where r is the crankshaft radius (half of the engine stroke), ω is the angular frequency, and λ is the crankshaft radius (r) divided with the connecting rod length (l) ratio (r/l).

MECHANICAL CONSTRAINTS

Symmetry boundary conditions are applied at the symmetry planes of the one-fourth model; that is, the deformations in the normal directions of these boundaries are constrained to zero (see Figure 3-2).

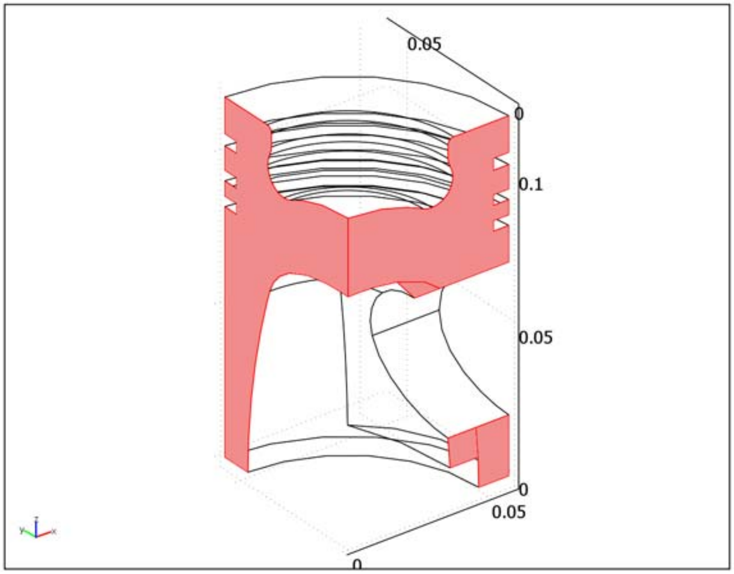


Figure 3-2: Symmetry boundaries.

The piston pin is assumed to be rigid in this model. This means that the boundaries at the pin hole (see Figure 3-3) must be contrained from moving in the positive normal direction. To include these contact boundary conditions to the model displacement dependent boundary forces are applied to the piston pin hole boundaries. These forces depend on the normal displacement of the pin hole boundaries in a very nonlinear way. A penalty/barrier method is used to model the contact forces F_c as follows

$$F_c = t_n + e_n \cdot g \qquad g > 0$$

$$F_c = t_n \cdot \exp\left(\frac{e_n}{t_n} \cdot g\right) \qquad g \le 0$$

where t_n is the input estimate of the contact force, e_n is the penalty stiffness, g is the gap, that is, the normal distance between the piston pin hole and the rigid pin.

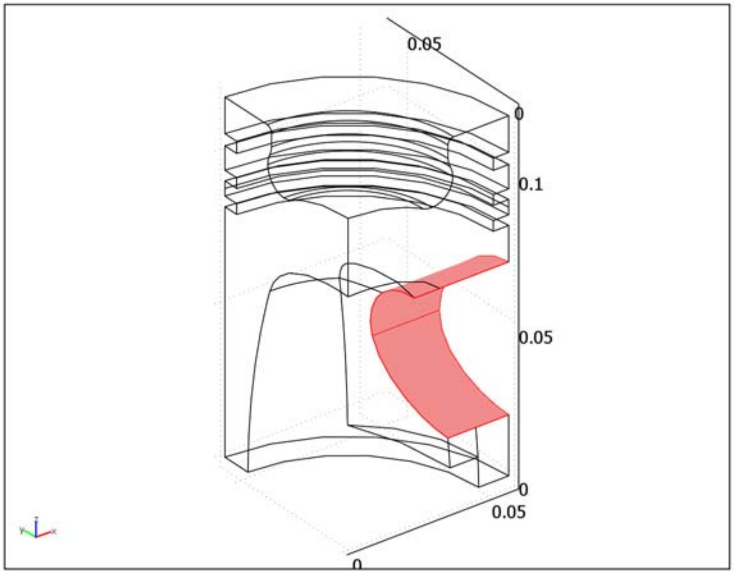


Figure 3-3: Contact boundary conditions at the pin hole.

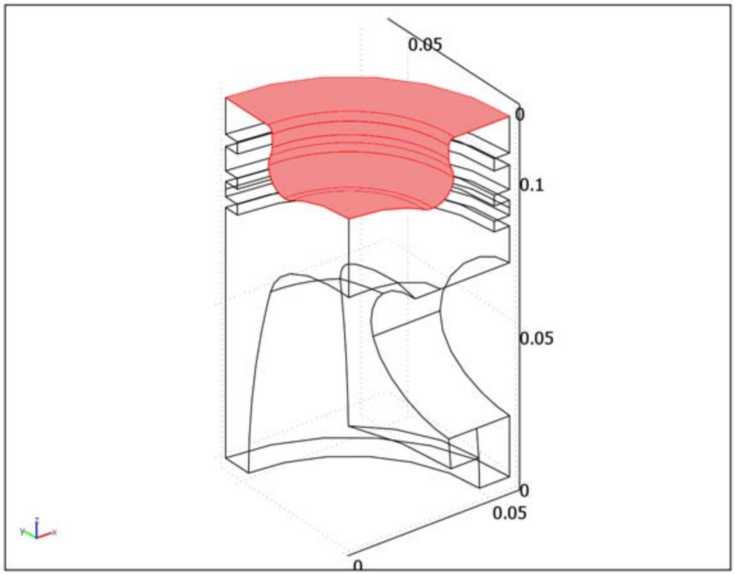
THERMAL BOUNDARY CONDITIONS

The effects of the cyclic swing in surface temperature during the combustion cycle are small compared to the time-averaged temperatures. The major effect of the heat transfer on thermal stresses are therefore taken into account by time-averaged boundary conditions (Ref. 1), that is, through constant convective boundary conditions.

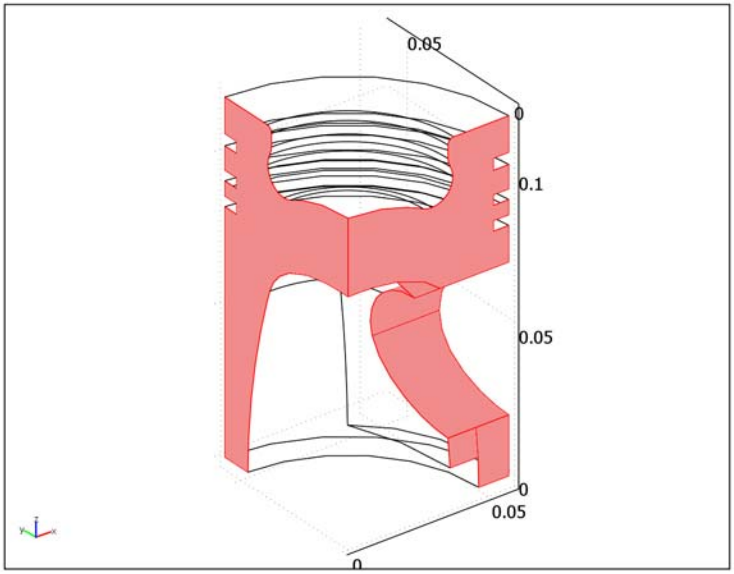
The heat transfer coefficients on all boundaries are some typical values for a high speed diesel engine, as well as the bulk combustion gas temperature, engine oil temperature, and cooling water temperatures (see Ref. 1).

The following thermal boundary conditions are applied:

- The combustion gas temperature (900 °C) is applied to the combustion bowl and piston crown areas as an external temperature. The heat transfer coefficient is set to 500 $W/(m^2 \cdot °C)$ in these areas.



- At the symmetry boundaries including the pin hole boundaries insulation/ symmetry conditions are applied.



- The outside of the piston is cooled by a cooling water whereas the inside is cooled by the engine oil, both at a temperature of 80 °C. Different heat transfer coefficients are applied on different boundaries and thereby reflecting the different cooling rates at each boundary. For example, a high heat transfer coefficient is applied to the bottom of the piston inside as this is the area where the piston oil cooling jet is directed.

Results and Discussion

The temperature distribution is depicted in Figure 3-4. The highest temperatures are located at the bowl edge as the figure clearly shows. High temperatures at the bowl edge lowers the fatigue limit of the material in this area.

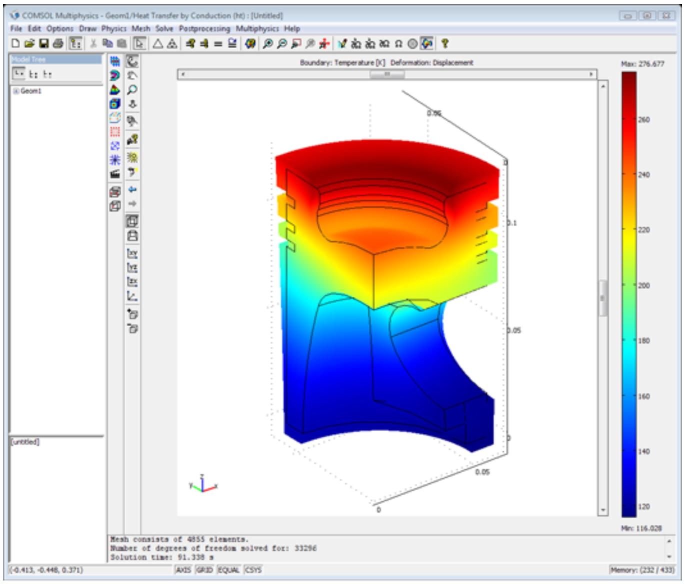


Figure 3-4: Temperature distribution.

Figure 3-5 shows the total displacement for the piston. The figure shows that the piston bends over the piston pin due to the combustion gas pressure. This bending action creates a high tensile stress in the bowl edge area above the pin hole.

The fact that the temperatures and the tensile stresses are high at the bowl edge can produce so-called high cycle fatigue cracks in this area.

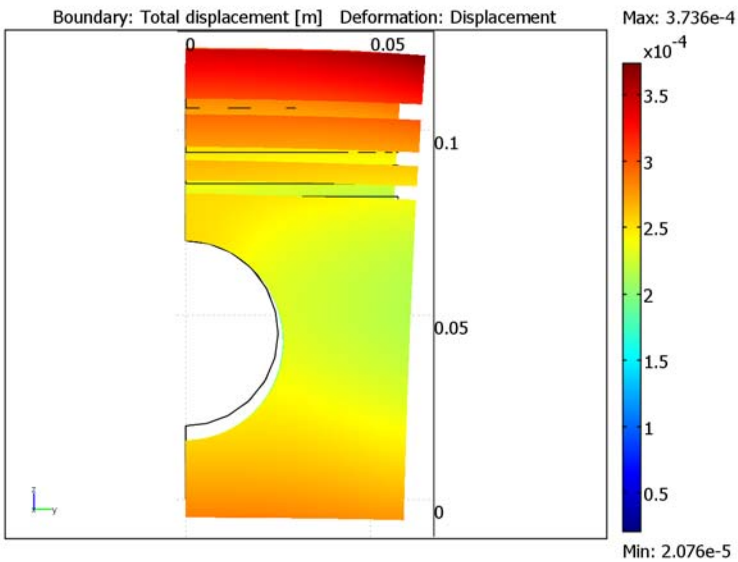


Figure 3-5: Total displacement.

Reference

1. Borman G., *Internal-Combustion Engine Heat Transfer*, Mechanical Engineering Department, University of Wisconsin-Madison, 1987.

Modeling in COMSOL Multiphysics

The nonlinear solver and the highly nonlinear option are used due to the contact boundary conditions at the piston pin hole (these boundary conditions result in very nonlinear boundary loads).

Model Library path: `Structural_Mechanics_Module/ Automotive_Applications/piston_3d`

Modeling Using the Graphical User Interface

1 Start COMSOL Multiphysics. This invokes the **Model Navigator**, which you can also open from an already running COMSOL Multiphysics session by choosing **New** from the **File** menu.

2 On the **New** page, select **3D** from the **Space dimension** list.

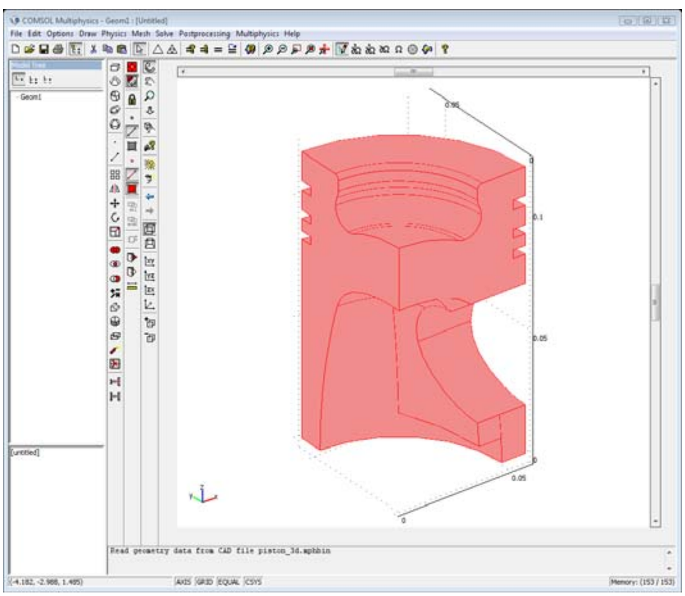
Note: To start a new model without any application modes from a running COMSOL Multiphysics session of the same space dimension you must first Ctrl-click to clear the current selection in the **Application Modes** list.

---

3 Click **OK** to close the **Model Navigator**. Note that the model does not include any physics modes yet; these will be added later on.

GEOMETRY MODELING

Select **Import>CAD Data From File** from the **File** menu and browse to the geometry file `piston_3D_geometry.mphbin` which is located in the folder `models/Structural_Mechanics_Module/Automotive_Applications/` in the COMSOL installation directory.



OPTIONS AND SETTINGS

1 Select **Constants** from the **Options** menu and enter the following constant names, expressions, and descriptions (optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
n	2000[1/min]	Revolutions per minute
omega	n*2*pi	Angular velocity
stroke	0.144[m]	Engine stroke
r	stroke/2	Crankshaft radius
conrod_length	0.26[m]	Connecting rod length
lda	r/conrod_length	
pistonacc	omega^2*r*(1+lda)	Piston acceleration
P	130e5[N/m^2]	Face load
tn	5e5[N/m^2]	Input estimate of contact force
en	1.0e14[N/m^3]	Penalty stiffness

2 From the **Options** menu select **Materials/Coefficients Library** and click **New**.

3 Enter then the following material parameters:

MATERIAL PARAMETER	EXPRESSIO N
E	70e9
nu	0.33
rho	2700
alpha	23e-6
C	944
k	236

4 Type `Aluminum` in the **Name** edit field click **OK** to close the **Materials/Coefficients Library** dialog box.

MODEL NAVIGATOR

1 Select **Model Navigator** from the **Multiphysics** menu.

2 Select **3D** from the **Space dimension** list.

3 In the application list double-click the **Structural Mechanics Module** folder, then select **Solid, Stress-Strain**, and finally click the **Add** button.

4 In the application list double-click first **COMSOL Multiphysics** and then **Heat Transfer**. Select **Conduction**, then click **Add**.

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

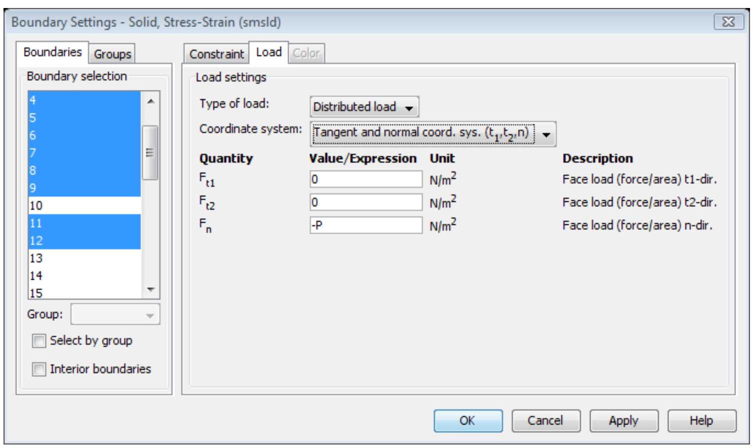
PHYSICS SETTINGS

Boundary Settings—Mechanical

1 Select **Solid, Stress-Strain (smsld)** from the **Multiphysics** menu.

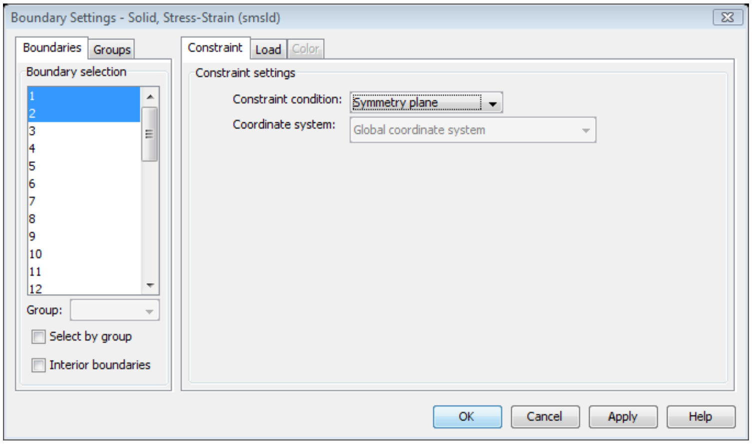
2 Enter the combustion pressure load on the **Load** page in the **Boundary Settings** dialog box for the Solid, Stress-Strain application mode according to the following table (see also Figure 3-1):

SETTINGS	BOUNDARIES 4–9, 11, 12, 27
Coordinate system	Tangent and normal coord. sys. (t_1,t_2,n)
F_n	-P



3 Enter the symmetry constraints on the **Constraint** page in the **Boundary Settings** dialog box.

SETTING	BOUNDARIES 1, 2, 29, 34, 35
Constraint condition	Symmetry plane



4 From the **Options** menu select **Expressions>Boundary Expressions**.

5 Enter the following **Boundary Expressions** variables on Boundaries 31 and 32. An additional boolean expression, `gap<=0`, is included in the exponent to avoid problems with numerical overflow.

NAME	EXPRESSION
`gap`	`nx*u+ny*v+nz*w`
`Fc`	`(gap>0)*(tn+en*gap)+(gap<=0)*tn*exp(gap*(gap<=0)*en/tn)`

6 You can now use the boundary expression variable `Fc` to enter the contact boundary condition to the pin hole.

7 Enter the following load expression to include a contact boundary condition on the **Load** page in the **Boundary Settings** dialog box.

SETTINGS	BOUNDARIES 31, 32
Coordinate system	Tangent and normal coord. sys. (t_1,t_2,n)
F_n	`-Fc`

Subdomain Settings—Mechanical

1 Select **Aluminum** from the **Library material** list on the **Material** page in the **Subdomain Settings** dialog box for the **Solid, Stress-Strain** application mode.

2 On the **Load** page enter the following inertia load.

SETTINGS	SUBDOMAIN 1
Coordinate system	Global coordinate system
F_z	`pistonacc*rho_smsld`

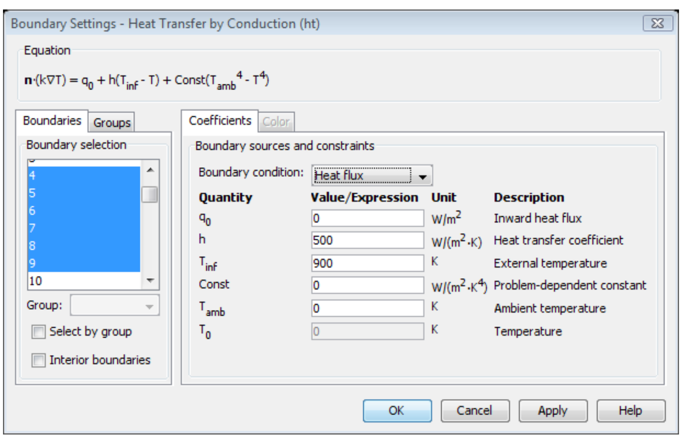
3 Select the **Include thermal expansion** check box and enter `T` in the **Temp** edit field and `25` in the **Tempref** edit field.

Boundary Settings—Thermal

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

2 Enter the combustion gas temperature and the associated heat transfer coefficients for the combustion bowl and piston crown area according to the following table in the **Boundary Settings** dialog box for the Heat Transfer by Conduction application mode.

SETTINGS	BOUNDARIES 4–9, 11, 12
Boundary condition	Heat flux
h	500
T_{inf}	900



3 Next set the thermal insulation boundary condition on the symmetry boundaries.

SETTING	BOUNDARIES 1, 2, 29, 31, 32, 34, 35
Boundary condition	Thermal insulation

4 The outer and inner surfaces that are cooled by the cooling water and the engine oil are given the following convective boundary conditions:

SETTINGS	BOUNDARIES 10, 13, 15, 18, 21, 23, 27, 28, 30, 33
Boundary condition	Heat flux
h	700
T_{inf}	80

5 The upper and inner areas of the piston rings have somewhat lower heat transfer coefficients compared to the surrounding areas.

SETTINGS	BOUNDARIES 14, 16, 17, 19, 20, 22
Boundary condition	Heat flux
h	500
T_{inf}	80

6 The ring lands, which are located between the ring grooves, have relatively low heat transfer coefficients.

SETTINGS	BOUNDARIES 24–26
Boundary condition	Heat flux
h	250
T_{inf}	80

7 Finally the area where the piston is cooled by the oil jet a high heat transfer coefficient is applied.

SETTINGS	BOUNDARY 3
Boundary condition	Heat flux
h	2400
T_{inf}	80

Subdomain Settings—Thermal

Select **Aluminum** from the **Library material** list on the **Physics** page in the **Subdomain Settings** dialog box for the Heat Transfer by Conduction application mode.

MESH GENERATION

1 Click the **Initialize Mesh** button.

COMPUTING THE SOLUTION

1 Open the **Solver Parameters** dialog box on the **Solve** menu.

2 Select **GMRES** from the **Linear system solver** list.

3 Select **Geometric multigrid** from the **Preconditioner** list.

4 On the **Stationary** page, select the **Highly nonlinear problem** check box.

5 Click **OK**.

6 Solve the problem by clicking the **Solve** button on the Main toolbar.

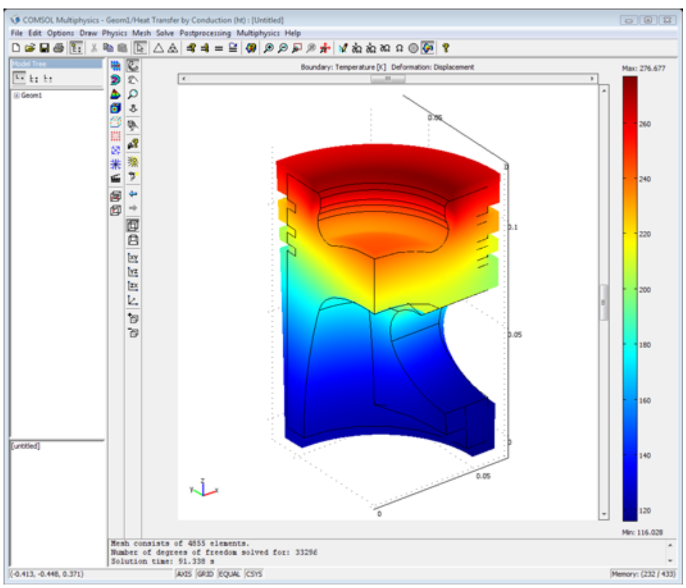
POSTPROCESSING AND VISUALIZATION

First create the temperature distribution plot.

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

2 Check **Deformed shape** plot and **Boundary** plot on the **General** page.

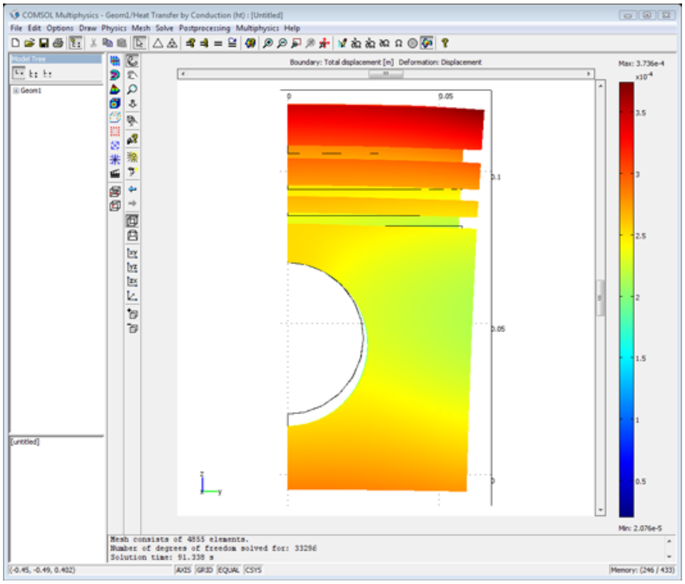
3 Click the **Boundary** tab. Select **Heat Transfer by Conduction (ht)>Temperature** from the **Predefined quantities** list and click **OK**.



The next plot shows the total displacement from a *YZ* view.

4 Still on the **Boundary** page, select **Solid, Stress-Strain (smsld)>Total displacement** from the **Predefined quantities** list and click **OK**.

5 Click the **Go to YZ View** button on the Camera toolbar.



Fuel Cell Bipolar Plate

This study presents a model that couples the thermal and structural analysis in a bipolar plate in a proton exchange membrane fuel cell (PEMFC). The fuel cell stack consists of unit cell of anode, membrane, and cathode connected in series through bipolar plates. The bipolar plates also serve as gas distributors for hydrogen and air that is fed to the anode and cathode compartments, respectively. Figure 3-6 below shows a schematic drawing of a fuel cell stack. The unit cell and the surrounding bipolar plates are shown in detail in the upper right corner of the figure.

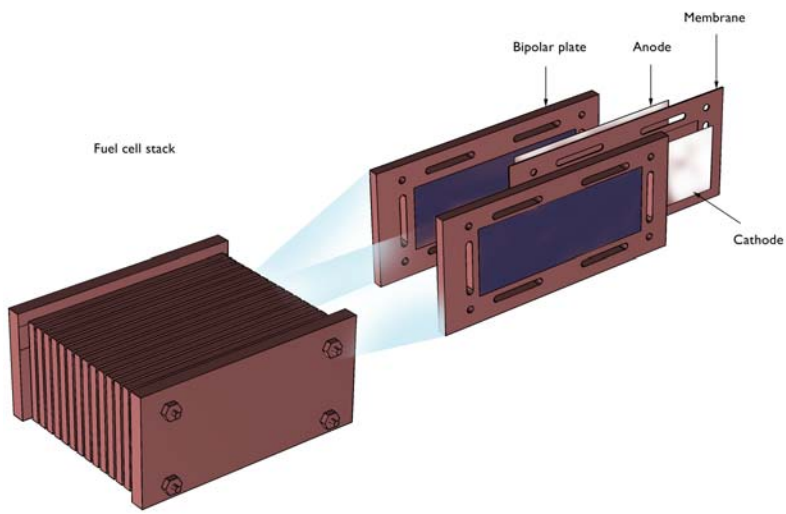


Figure 3-6: Schematic drawing of the fuel cell stack. The unit cell consists of an anode, membrane electrolyte, and a cathode. The unit cell is supported and supplied with reactants through the bipolar plates. The bipolar plates also serve as current collectors and feeders.

The PEMFC is one of the strongest alternatives for automotive applications where it has the potential of delivering power to a vehicle with a higher efficiency, from oil to propulsion, than the internal combustion engine.

The fuel cell operates at temperatures just below 100 °C, which means that it has to be heated at start-up. The heating process induces thermal stresses in the bipolar plates. The analysis in this model reveals the magnitude and nature of these stresses.

Model Definition

Figure 3-7 below show the detailed model geometry. The plate consists of gas slits that form the gas channels in the cell, holes for the tie rods that keep the stack together, and the heating elements, which are positioned in the middle of the gas feeding channel for the electrodes. Due to symmetry, it is possible to reduce the model geometry to 1/8 of the actual size of the cell.

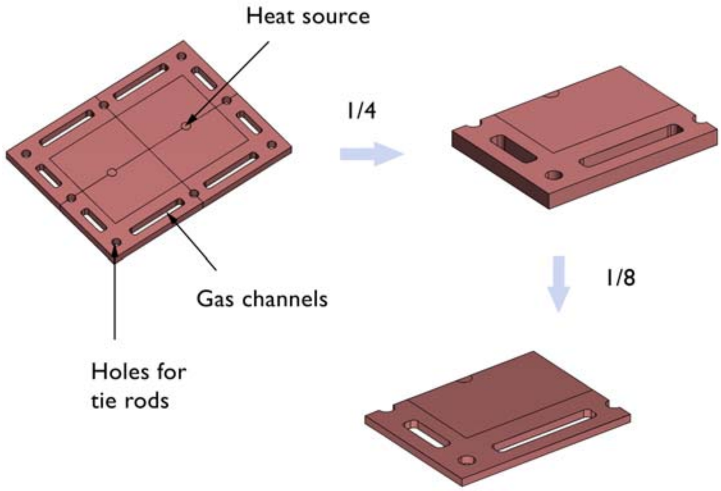


Figure 3-7: The modeled geometry.

The study consists of a thermal analysis and a structural analysis, both for steady operation. In the following specification of the problem a number of constants are used; their values are listed in the table below. The constant Q_1 represents a power of 200 W distributed over the hole through the 25 bipolar plates in the fuel cell stack.

CONSTANT	EXPRESSION	DESCRIPTION	UNITS
Q_1	`200/25/(pi*0.005^2*0.01)`	Volume power in one heating element	W/m^3
P_1	`9.82e5`	Pressure on the active part	N/m^2
P_2	`7.85e6`	Pressure on the manifold	N/m^2
h_1	5	Heat transfer coefficient to the surroundings	W/(m^2·K)
h_2	50	Heat transfer coefficient in the gas channels	W/(m^2·K)

THERMAL ANALYSIS

The general equation is the steady-state heat equation according to:

$$\nabla \cdot (-k\nabla T) + Q = 0 \tag{3-1}$$

where k denotes the thermal conductivity of the different materials, T the temperature, and Q a heat source or heat sink.

The modeled domain consists of three different subdomains. The first subdomain corresponds to the active part of the cell, where the electric energy is produced and the current is conducted. The production and conduction of current involve some losses. It is assumed in this model that the cell is heated prior to operation. This means that the model does not account for the heat sources due to the production and conduction of current; see Figure 3-8.

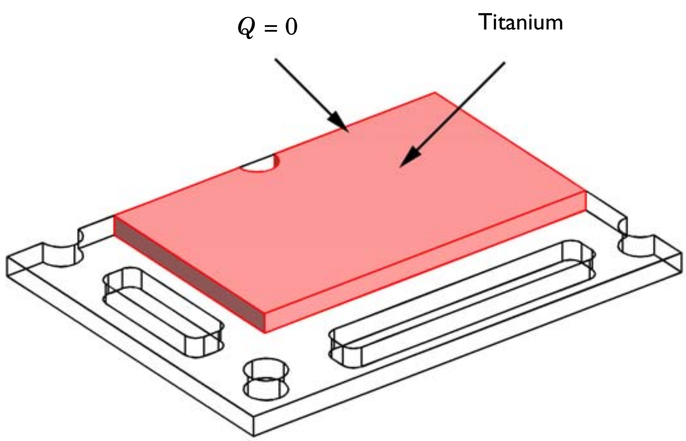


Figure 3-8: The active part of the cell bipolar plate is made of titanium.

The second subdomain corresponds to the heating element in the bipolar plate, which is made of aluminum. The power from this element is assumed to be uniformly distributed in the small cylindrical subdomain; see Figure 3-9.

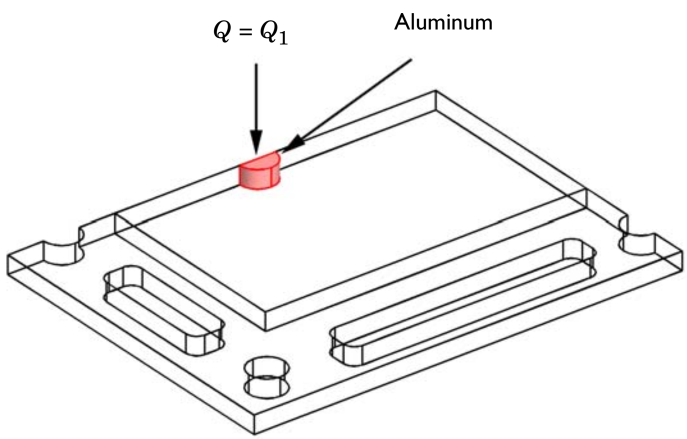


Figure 3-9: The heating element in the bipolar plate is made of aluminum.

The last subdomain forms the manifold in the cell. In this subdomain, only heat conduction is present. The construction material is titanium.

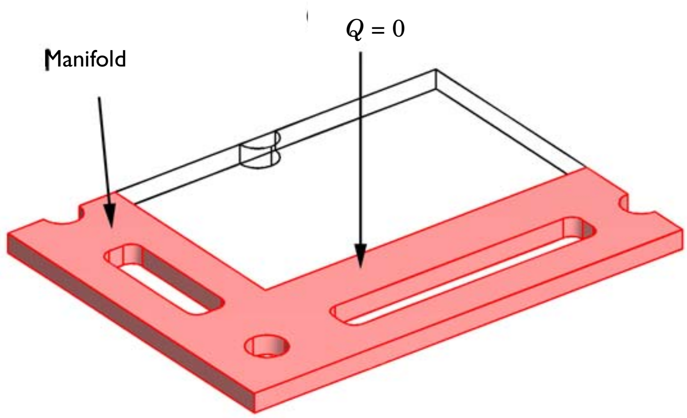


Figure 3-10: The subdomain that forms the manifold of the cell.

The thermal conductivity of the materials and the magnitude of the heat source form the input data needed in the model. You can find the data in the COMSOL Multiphysics file for this model.

The boundary conditions for the heat transfer analysis are symmetry conditions and convective conditions. The symmetry condition states that the flux is zero perpendicular to the boundary.:

$$(-k\nabla T)\cdot\mathbf{n} = 0. \tag{3-2}$$

The convective conditions set the heat flux proportional to the temperature difference between the fluid outside and the temperature at the boundary. The heat transfer coefficient is the proportionality constant:

$$(-k\nabla T)\cdot\mathbf{n} = h(T-T_{\mathrm{fluid}}). \tag{3-3}$$

In this equation, h denotes the heat transfer coefficient.

Figure 3-11 below shows the insulation boundaries.

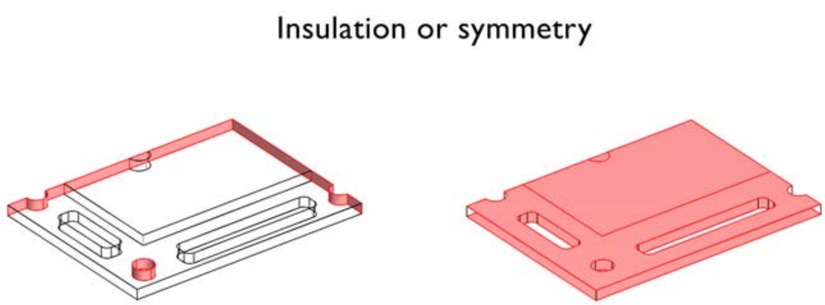


Figure 3-11: Insulation boundaries.

Figure 3-12 shows the convective boundaries. The heat transfer coefficient is different for the different groups of boundaries in the figure.

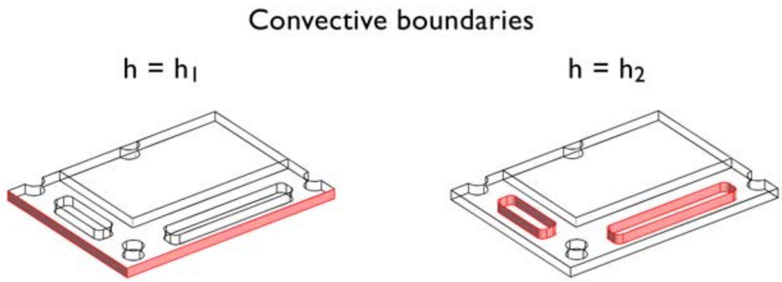


Figure 3-12: The convective boundaries.

STRUCTURAL ANALYSIS

The material properties for titanium and aluminum are obtained from COMSOL Multiphysics' materials database. The only volume loads present in the domain are those generated by thermal expansion. The thermal loads are proportional to the temperature difference between the reference state and the actual temperature.

The loads and constraints on the boundaries are shown in Figure 3-13 below. There are no displacements in the direction perpendicular to the symmetry boundaries.

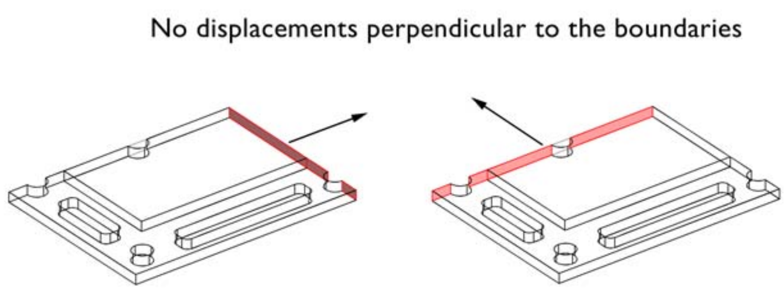


Figure 3-13: The symmetry boundaries.

The base surface of the plate is also a symmetry boundary, which implies that there are no displacements in the *z* direction at the position of the boundary.

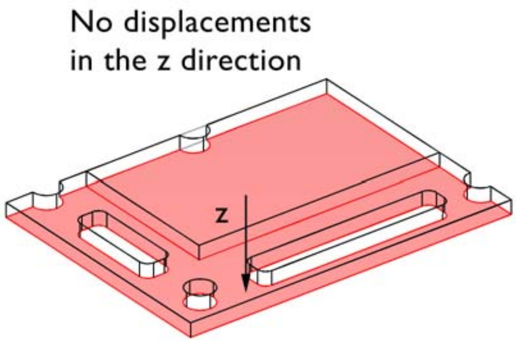


Figure 3-14: No displacement perpendicular to the symmetry plane.

In addition, the load applied by the tie rods on the stack is applied in the manifold and the active part at the bipolar plate. The pressure is different in these two subdomains.

The loads and constraints on all other boundaries are assumed to be negligible. In this context, the only questionable assumption is the possible loads and constraints imposed by the tie rods on the holes in the bipolar plate. In a more detailed analysis, it would be possible to couple a model for the tie rods with a detailed model of the bipolar plate.

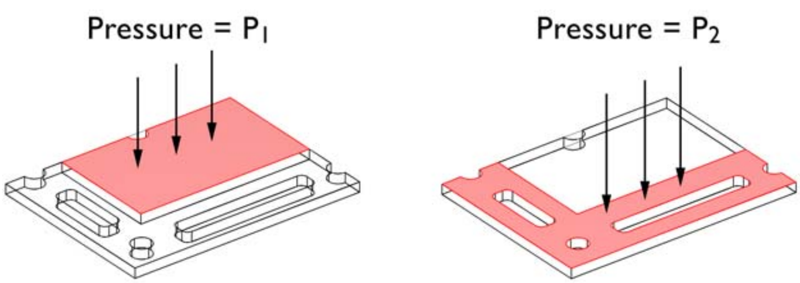
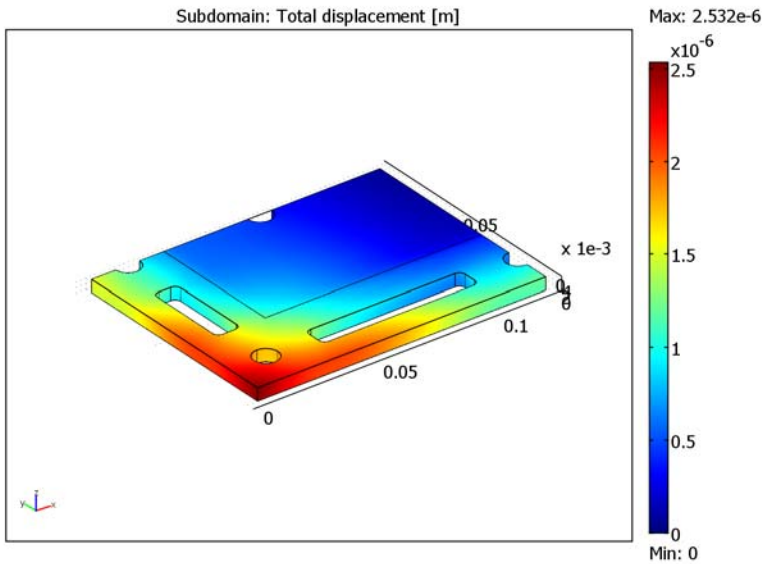


Figure 3-15: Pressure applied externally.

Results

To estimate the influence of the temperature on the displacements and stresses and strains in the plate, you can solve the problem for the structural part first, assuming constant temperature at the reference state. Figure 3-16 shows the displacement and the von Mises stresses in the plate. As you can see from the plots, the von Mises effective stresses are relatively large in the manifold region of the plate.



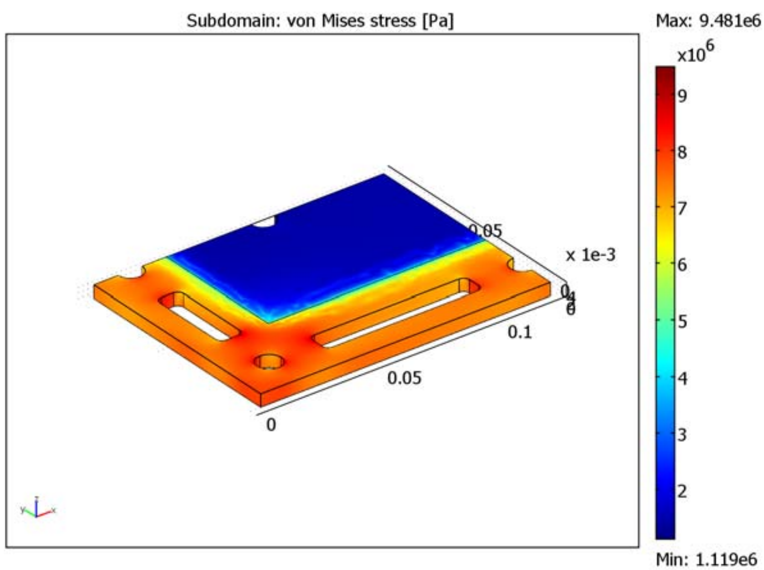


Figure 3-16: The displacement in the plate (top) and the von Mises stresses caused by the external pressure applied by the end plates and the tie rods.

Figure 3-17 shows the temperature field at steady state. The slits for the gas present the largest sinks of heat at the boundaries. The temperature decreases almost radially with the distance from the heating element.

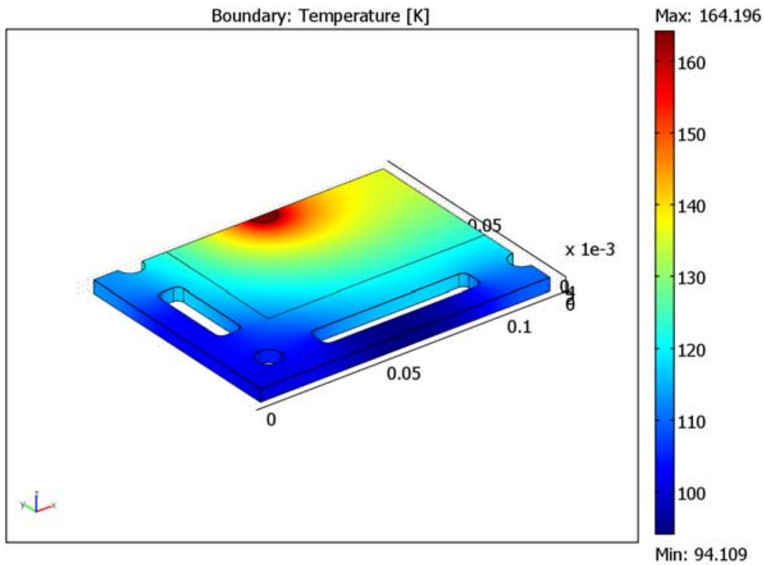


Figure 3-17: Temperature distribution in the plate.

The displacements and stresses generated due to the external pressure and the temperature gradients are shown in Figure 3-18.

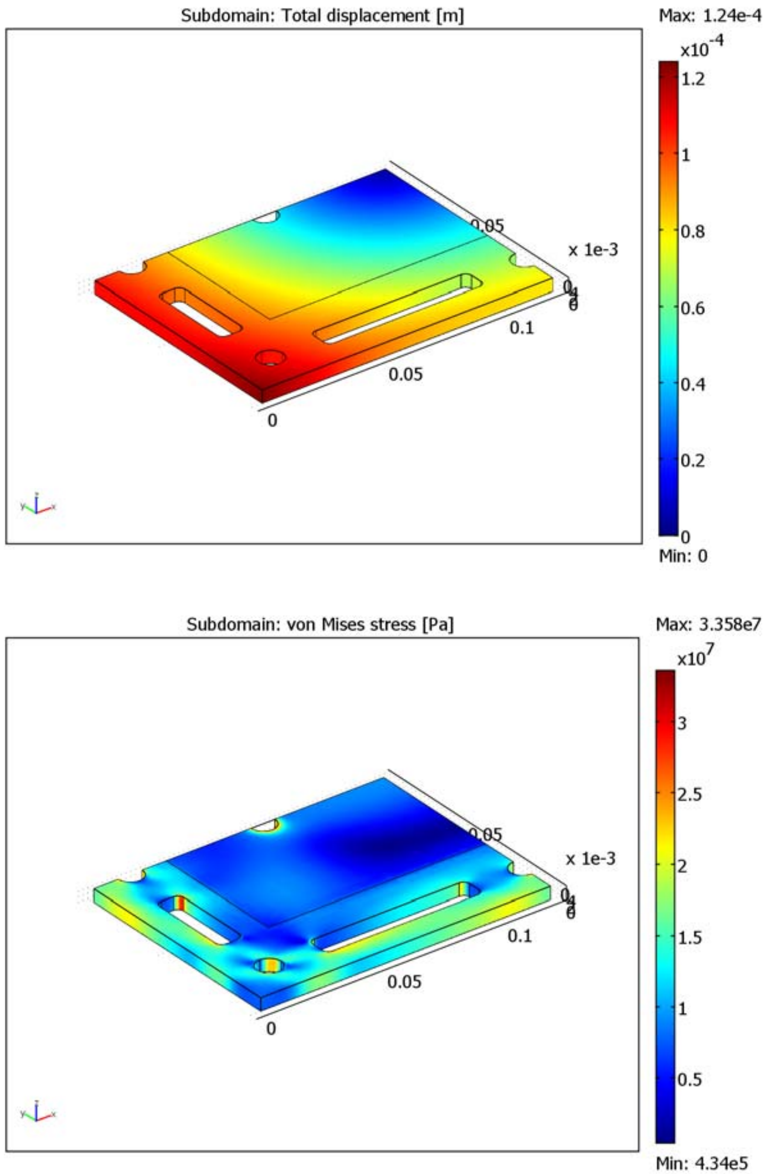


Figure 3-18: Displacements (top) and von Mises stresses due to temperature and external loads.

Figure 3-18 shows that the thermal loads generate stresses that are one order of magnitude larger than those generated by the external pressure loads. The loads are far from the critical values, but the displacements can be even more important. The displacements should be small enough to grant that the membrane can fulfill its function in separating hydrogen and oxygen in the anode and cathode compartments, respectively.

Model Library path: `Structural_Mechanics_Module/Automotive_Applications/bipolar_plate`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Click the **New** button or start COMSOL Multiphysics to open the **Model Navigator**.

2 Select **3D** from the **Space dimension** list.

3 Click the **Multiphysics** button.

4 Open the **Structural Mechanics Module** folder and then **Solid, Stress-Strain**. Select the **Static analysis** and click the **Add** button to add the application mode.

5 Open the **COMSOL Multiphysics>Heat Transfer>Conduction** folder. Select **Steady-state analysis** and click **Add** to add the heat transfer application mode.

6 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, select **Constants**.

2 Enter the following constant names, expressions, and (optionally) descriptions. When finished, click **OK**.

NAME	EXPRESSION	DESCRIPTION
P1	9.82e5[Pa]	Pressure on the active part
P2	7.856e6[Pa]	Pressure on the manifold

GEOMETRY MODELING

Create the geometry by extruding a 2D triangular mesh, resulting in prism elements.

1 Select **Work-Plane Settings** from the **Draw** menu to open the **Work-Plane Settings** dialog box, and click **OK** to create a default *x*-*y* work plane.
2 Choose **Options>Axes/Grid Settings**. On the **Grid** page, clear **Auto**, and set axis and grid settings in Geom2 according to the following table. Click **OK**.

AXIS		GRID	
x min	`-0.01`	x spacing	`2.5e-3`
x max	`0.15`	Extra x	
y min	`-0.01`	y spacing	`2.5e-3`
y max	`0.1`	Extra y	

3 Draw a rectangle with opposite corners at (0, 0) and (0.12, 0.09).
4 Draw a rectangle with opposite corners at (0.03, 0.035) and (0.12, 0.09).
5 Draw three centered circles with radius 0.005 and center at (0.015, 0.015), (0.015, 0.09), and (0.12, 0.015).
6 Draw a rectangle with opposite corners at (0.01, 0.035) and (0.02, 0.07).
7 Draw a rectangle with opposite corners at (0.035, 0.01) and (0.105, 0.02).

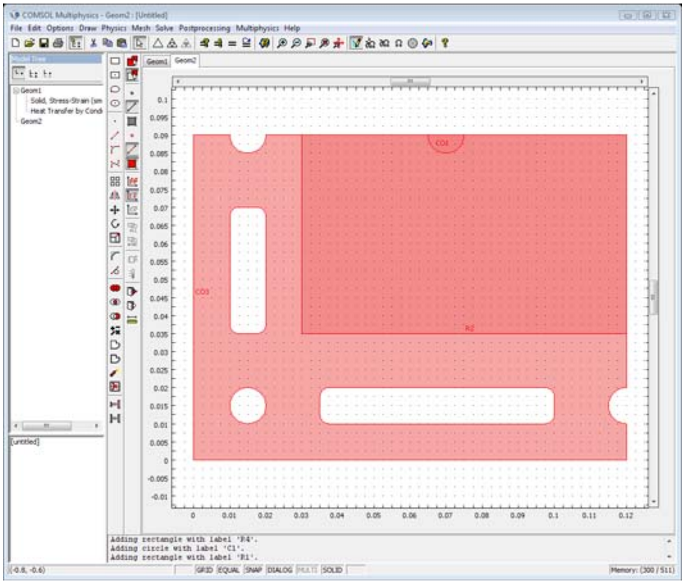
Create fillets on the two last created rectangles.

8 Click the **Fillet/Chamfer** toolbar button to open the **Fillet/Chamfer** dialog box.
9 Select all vertices in the two rectangle by clicking in the drawing area or in the **Vertex selection** list.
10 Type `2.5e-3` in the **Radius** edit field, then click **OK** to create the fillets.
11 Select all objects except the second largest rectangle (R2) and click the **Difference** button on the Draw toolbar to cut out the holes.

Create the subdomain where the heating is done.

12 Draw a centered circle with radius 0.005 and center at (0.07, 0.09).
13 Draw a rectangle with opposite corners at (0.065, 0.09) and (0.075, 0.095).

14 Select both the circle and the rectangle, then click the **Difference** button.



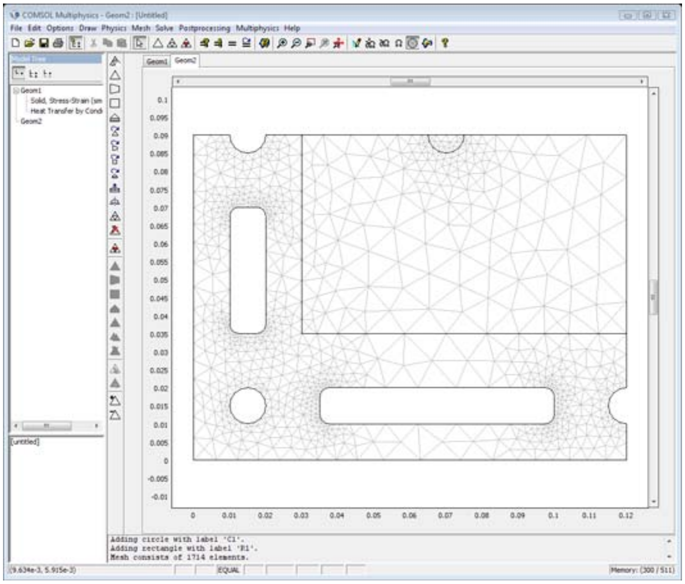
You are now finished with the drawing of the 2D geometry. Create the 3D geometry by extruding the 2D mesh. To do this you must first create the 2D mesh.

MESH GENERATION

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

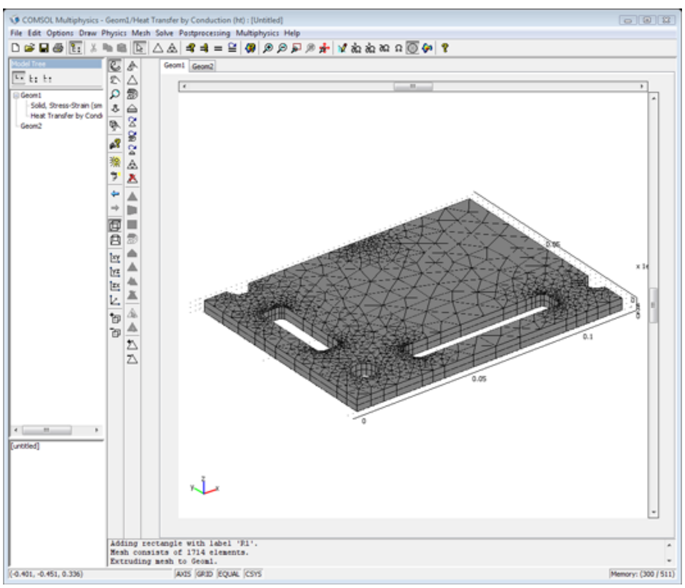
2 Select **Coarse** from the **Predefined mesh sizes** list.

3 Click the **Remesh** button to mesh the 2D geometry and click **OK**.



4 Choose **Extrude Mesh** from the **Mesh** menu to open the **Extrude Mesh** dialog box.

5 Type `0.005` in the **Distance** edit field, then click **OK** to create the extruded mesh.



PHYSICS SETTINGS

Boundary Conditions

Start with the structural boundary conditions.

1 Choose **Geom1: Solid, Stress-Strain (smsld)** from the **Multiphysics** menu.

2 Choose **Physics>Boundary Settings** and enter the boundary settings according to the following table; when done, click **OK**.

	BOUNDARIES 3, 5, 20, 23, 25, 36, 42–44		BOUNDARY 24		BOUNDARY 4	
Page	Constraint		Load		Load	
	Constraint condition	Symmetry plane	F_z	-P1	F_z	-P2

Continue with the heat transfer boundary conditions:

3 Choose **Geom1: Heat Transfer by Conduction (ht)** from the **Multiphysics** menu to set the boundary conditions for the heat transfer application mode.

4 Choose **Physics>Boundary Settings** and enter the boundary settings according to the following table; when done, click **OK**.

SETTINGS	BOUNDARIES 1, 2	BOUNDARIES 8–10, 12, 13, 17–19, 26–30, 37–39
Boundary condition	Heat flux	Heat flux
h	5	50
T_{inf}	20	80

Subdomain Settings

Define the material properties using the materials library. Start with the structural mechanics application mode.

1 Choose **Geom1: Solid, Stress-Strain (smsld)** from the **Multiphysics** menu.

2 Choose **Physics>Subdomain Settings**.

3 Select Subdomains 1 and 2 from the **Subdomain selection** list. Click the **Load** button on the **Material** page to open the **Materials/Coefficients Library** dialog box.

4 Select **Titanium beta-21S** from the **Materials** tree and click **OK** to close the **Materials/Coefficients Library** dialog box. Click **OK** to close the **Subdomain Settings** dialog box.

5 Select Subdomain 3 from the subdomain list and clear the **Active in this domain** check box to disable the structural application mode in this domain.

6 Choose **Geom1: Heat Transfer by Conduction (ht)** from the **Multiphysics** menu to change to the heat transfer application mode.

7 Choose **Physics>Subdomain Settings**.

8 Select Subdomains 1 and 2 from the **Subdomain selection** list. Click the **Load** button on the **Material** page to open the **Materials/Coefficients Library** dialog box.

9 Select **Titanium beta-21S** from **Model** in the **Materials** tree and click **OK** to close the **Materials/Coefficients Library** dialog box.

10 Select Subdomain 3 from the **Subdomain** list. Click the **Load** button on the **Material** page to open the **Materials/Coefficients Library** dialog box.

11 Select **Aluminum** from the **Materials** tree and click **OK** to close the **Materials/Coefficients Library** dialog box.

12 Type `200/25/(pi*0.005^2*0.01)` as the heat source in the **Q** edit field. Click **OK** to close the **Subdomain Settings** dialog box.

COMPUTING THE SOLUTION WITHOUT THERMAL EXPANSION

In the first analysis, solve the structural part without thermal expansion.

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

2 On the **Solve For** page select **Solid, Stress-Strain (smsld)** from the **Solve for variables** list and click **OK** to close the dialog box.

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

The structural mechanics problem without thermal expansion is now solved.

POSTPROCESSING AND VISUALIZATION

Look at the total displacement and von Mises stresses.

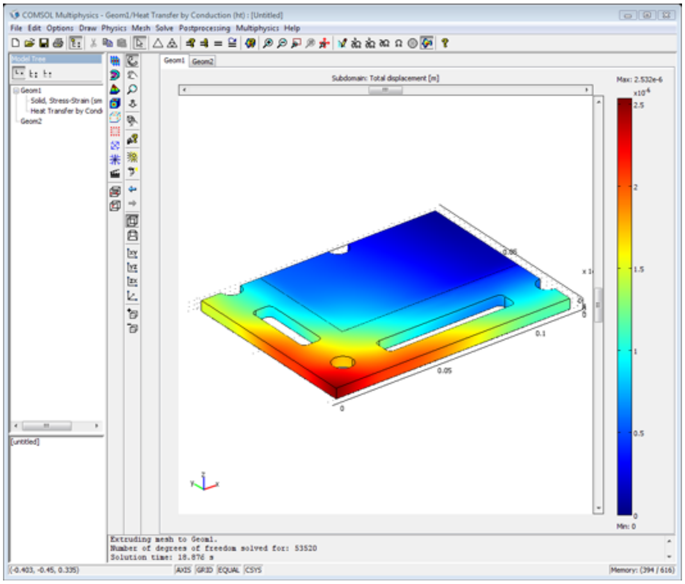
1 Choose **Plot Parameters** from the **Postprocessing** menu.

2 Select the **Subdomain** and **Geometry edges** check boxes in the **Plot type** area on the **General** page.

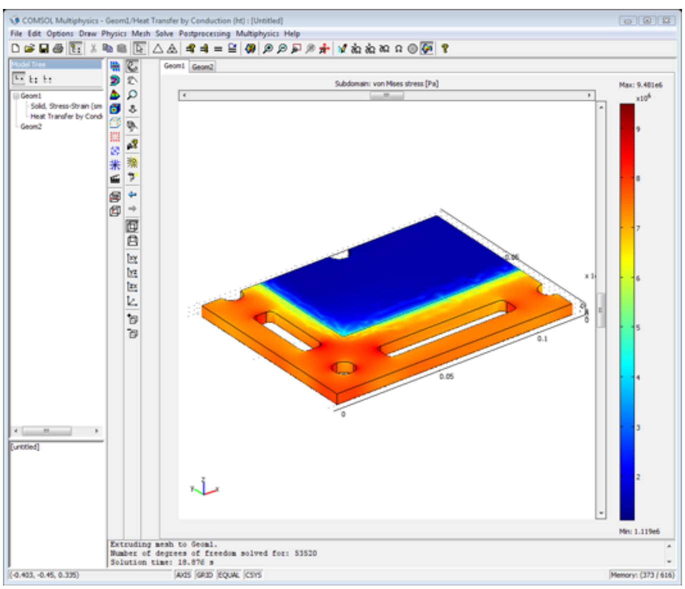
3 Click the **Subdomain** tab.

4 From the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>Total displacement**.

5 Click **Apply**.



6 From the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>von Mises stress**, then click **Apply**.



COMPUTING THE SOLUTION WITH THERMAL EXPANSION

Start to calculate the temperature field.

1 Open the **Solver Manager** dialog box.
2 Select **Heat Transfer by Conduction (ht)** from the **Solve for variables** list on the **Solve For** page, then click **OK** to close the dialog box.
3 Click the **Solve** button on the Main toolbar to solve for the temperature.

   Next, specify the temperature load in the structural mechanics application mode.

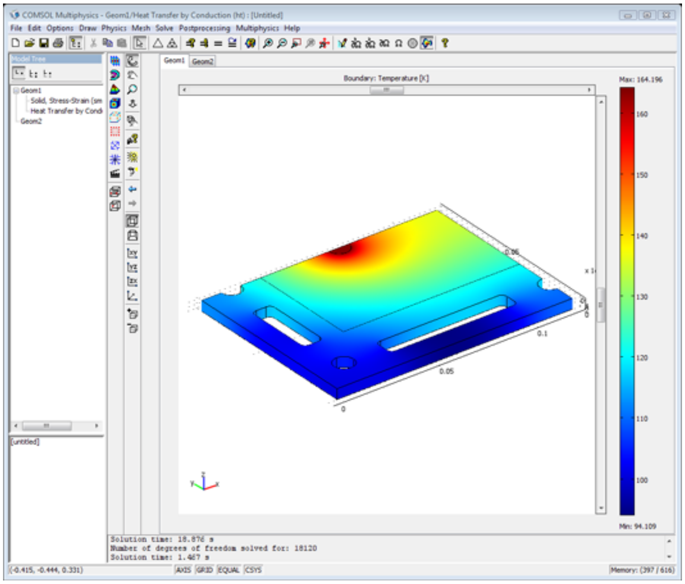
4 Choose **Geom1: Solid, Stress-Strain (smsld)** from the **Multiphysics** menu.
5 Choose **Physics>Subdomain Settings**.
6 Select Subdomains 1 and 2 from the **Subdomain selection** list.
7 Click the **Load** tab.
8 Select the **Include thermal expansion** check box and type `T` in the **Temp** edit field.
9 Open the **Solver Manager** dialog box and click the **Initial value** tab.
10 Click the **Stored solution** option button in the **Values of variables not solved for and linearization point** area.
11 Click the **Store Solution** button to save the temperature solution.
12 Select **Solid, Stress-Strain (smsld)** from the **Solve for variables** list on the **Solve For** page and click **OK** to close the dialog box.
13 Click the **Solve** toolbar button to solve the structural mechanics application mode including thermal expansion.

POSTPROCESSING AND VISUALIZATION

Visualize the temperature field, total displacement, and von Mises stress level including thermal expansion.

1 From the **Postprocessing** menu, select **Plot Parameters**.
2 Select the **Boundary** check box and clear the **Subdomain** check box in the **Plot type** area on the **General** page to clear the **Subdomain** plot type.
3 Click the **Boundary** tab.
4 Select **Heat Transfer by Conduction (ht)>Temperature** from the **Predefined quantities** list.

5 Click **Apply**.

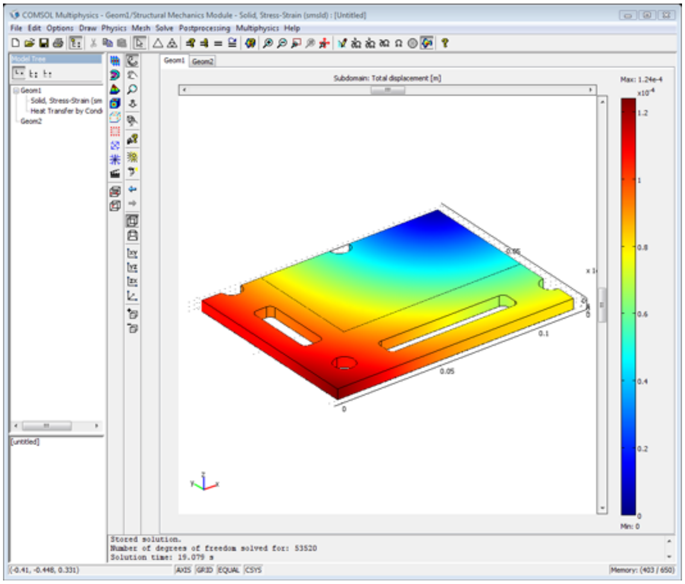


6 Select the **Subdomain** and **Geometry edges** check boxes in the **Plot type** area on the **General** page.

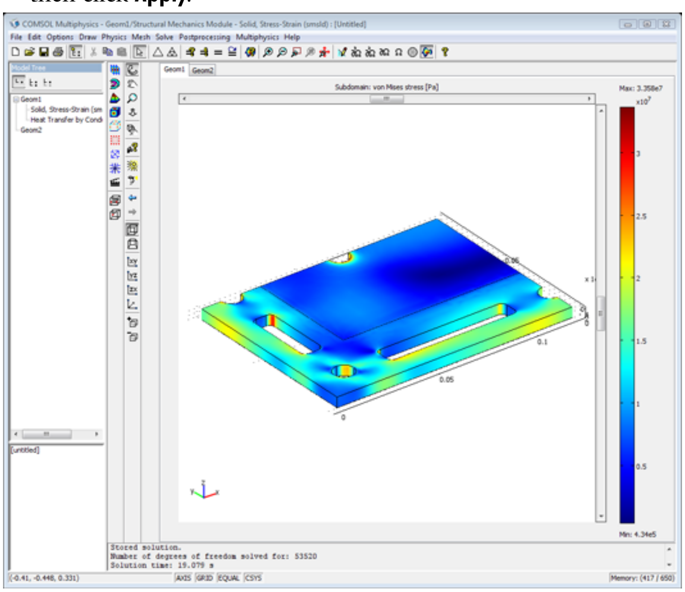
7 Click the **Subdomain** tab.

8 Select **Solid, Stress-Strain (smsld)>Total displacement** from the **Predefined quantities** list.

9 Click **Apply**.



10 Select **Solid, Stress-Strain (smsld)>von Mises stress** from the **Predefined quantities** list, then click **Apply**.



11 Click the **Head Light** toolbar button, then click the **Zoom Extents** toolbar button.

Spinning Gear

Introduction

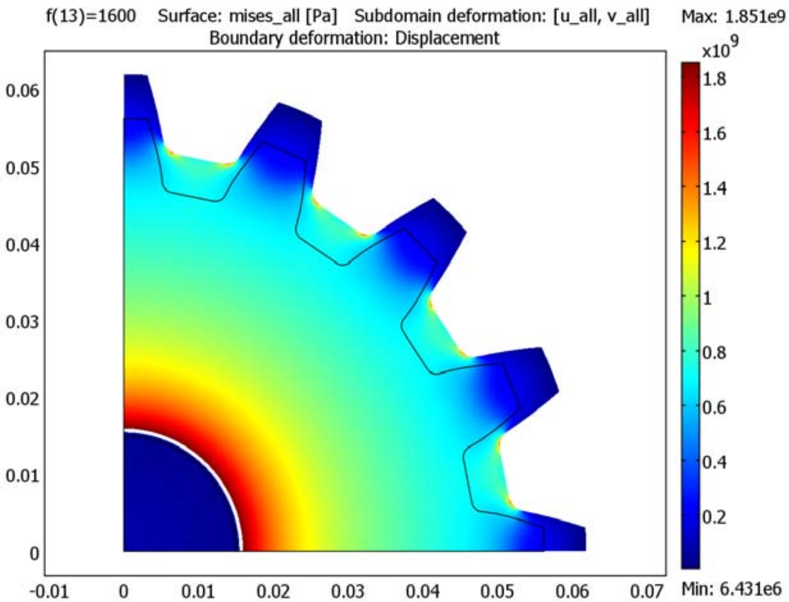


Figure 3-19: Stresses and deformation at 1600 Hz.

One way to fasten a gear to a shaft is by thermal interference. In preparation of the assembly, the shaft diameter is oversized and the gear thermally expanded in a heat treating oven. At an appropriate expansion state, the gear is removed from the oven, slid onto the shaft, and allowed to cool. As the gear temperature drops, the gear shrinks and comes into contact with the shaft before it reaches its original shape. From this point on, additional gear shrinkage results in hoop stresses in the gear as well as normal compression of the shaft. At thermal equilibrium, an intimate bond between the two components is reached.

Such an assembly can operate safely in many situations. However, there are operating conditions under which the fastening stresses become insufficient—for instance, when spinning the assembly at high rpm.

The goal of this analysis is to determine the critical spinning frequency at which gear and shaft separate.

Model Definition

The model computations consist of two steps:

- Thermal interference fit
  - Import the gear geometry from a given CAD file and draw the shaft using COMSOL Multiphysics solid modeling tools.
  - Fasten the gear to the shaft by thermal interference: Initially, both shaft and gear reside at room temperature (23 °C). Then, the gear is heated to 700 °C, positioned on the shaft, and allowed to cool.
- Spinning the shaft-gear assembly
  - Spin the shaft-gear assembly and determine the separation frequency.

Assume plane stress conditions for all computations and neglect contact phenomena during separation.

GEOMETRY

The geometry in Figure 3-20 consists of a shaft and a gear.

Shaft specifications:

- Material: Steel AISI 4340
- Radius: 0.015 m
- Length: 0.1 m

Gear specifications:

- CAD file: `gear.dxf`. This file is included in the model folder and was taken from Ref. 1.
- Material: Steel AISI 4340

- Thickness: 0.01 m

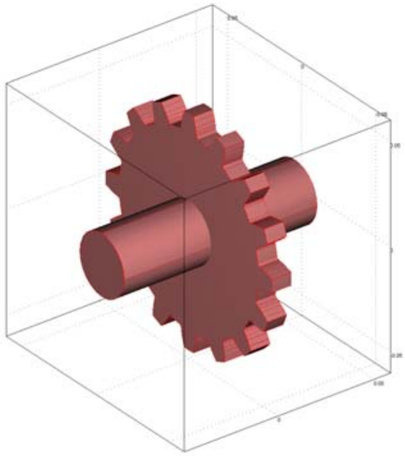


Figure 3-20: Gear geometry

DOMAIN EQUATIONS

The given problems are solved by computing the stress and deflection fields of the steady thermal interference and critical separation states. Starting with the stress-strain relation

$$\begin{bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{bmatrix} = D\begin{bmatrix}\varepsilon_x\\ \varepsilon_y\\ \gamma_{xy}\end{bmatrix}_{\mathrm{el}}$$

and the thermal strain relation

$$\begin{bmatrix}\varepsilon_x\\ \varepsilon_y\\ \gamma_{xy}\end{bmatrix}_{\mathrm{el}} = \begin{bmatrix}\varepsilon_x\\ \varepsilon_y\\ \gamma_{xy}\end{bmatrix} - \begin{bmatrix}\alpha_t\\ \alpha_t\\ 0\end{bmatrix}(T - T_{\mathrm{ref}}) \tag{3-4}$$

where the subscript "el" stands for elastic and α_t is the coefficient of thermal expansion, you can state that

$$\begin{bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{bmatrix} = D\left(\begin{bmatrix}\varepsilon_x\\ \varepsilon_y\\ \gamma_{xy}\end{bmatrix} - \begin{bmatrix}\alpha_t\\ \alpha_t\\ 0\end{bmatrix}(T - T_{\mathrm{ref}})\right) + \begin{bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{bmatrix}_{\mathrm{res}}$$

where

$$D = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

For more information about the underlying equations for the Plane Stress application mode, see the *Structural Mechanics Module User's Guide*.

In the second part of the analysis, the forcing term f represents the centripetal body force:

$$f = \begin{bmatrix} F \cdot \cos(\theta) \\ F \cdot \sin(\theta) \end{bmatrix}, F = \rho\omega^2 r = 4\pi^2 f^2 \rho \sqrt{x^2 + y^2}$$

BOUNDARY CONDITIONS

To prevent rigid body translation and rotation, you must impose some constraints (Dirichlet conditions): For computational efficiency, the analysis only includes a symmetric quarter of the geometry. By setting the normal displacements on the symmetry boundaries to zero, it is easy to constrain the model.

Modeling in COMSOL Multiphysics

Because the analysis neglects the contact phenomena, the gear geometry is modeled at the thermal expansion of 700 °C, at which it fits precisely on the shaft. The model assumes that the gear expands freely in the heat-treating oven and that the heating profile removes all internal stresses. When the assembly is spun, the gear expands more quickly than the shaft and reaches a critical separation point.

Results

INTERFERENCE

In the first analysis step, you obtain the stress distribution of the thermal interference. Figure 3-21 illustrates the hoop stresses in the gear which increase gradually toward the interface between shaft and gear. As a result, the shaft is exposed to normal compression.

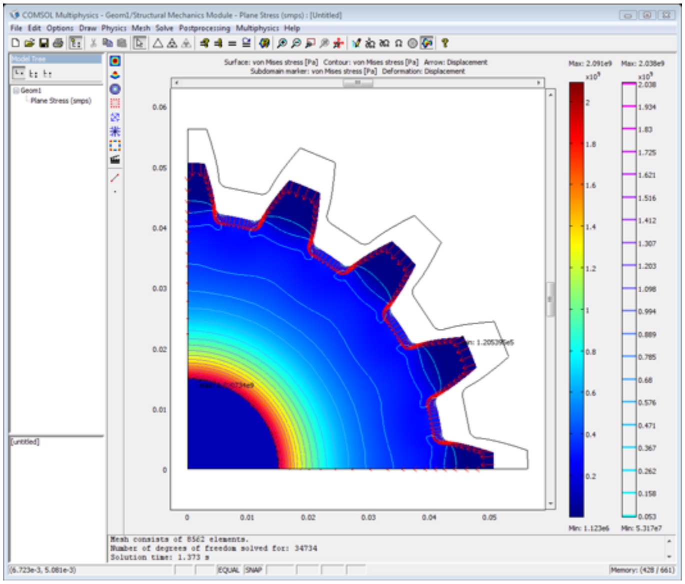


Figure 3-21: Von Mises stresses superimposed on shaft and gear. Note the hoop stresses.

SEPARATION

The parametric analysis spins the prestressed assembly at various frequencies, and you can plot the displacement between the shaft and the gear. Figure 3-22 and Figure 3-23 illustrate an advanced displacement state at 1600 Hz and a displacement versus frequency plot, respectively. The separation frequency occurs at the minimum of about 1550 Hz (Figure 3-23).

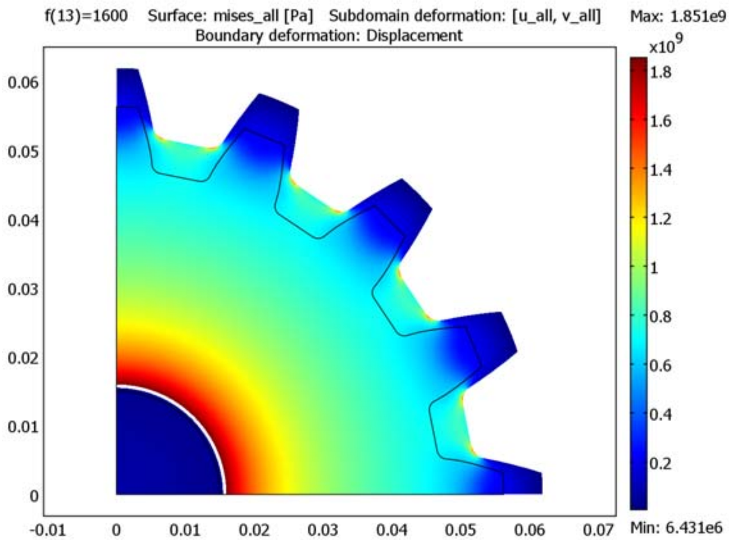


Figure 3-22: Von Mises stresses and deflection at 1600 Hz.

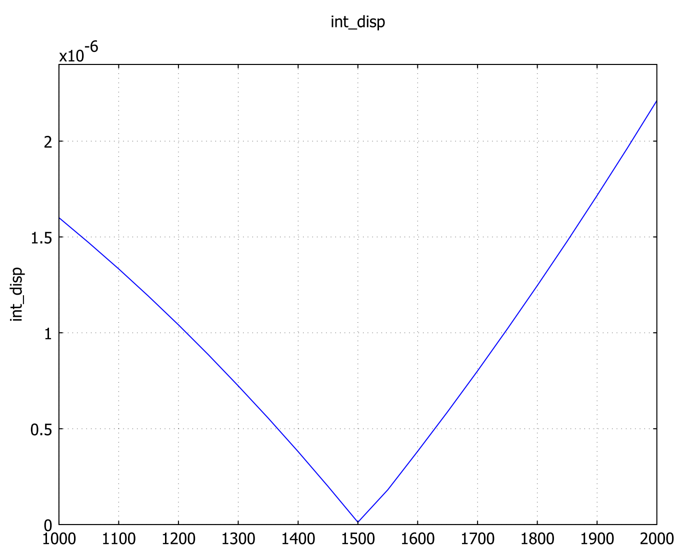


Figure 3-23: Displacement vs. frequency; separation occurs at 1550 Hz.

SCRIPTING AND OPTIMIZATION

The above frequency sweep was obtained by trial and error. If COMSOL Script or MATLAB is available, you can obtain the separation frequency by optimization (Figure 3-24).

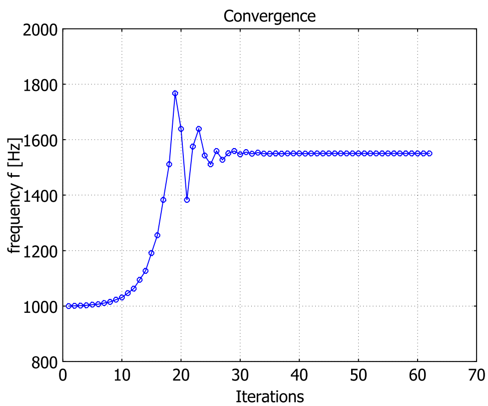


Figure 3-24: Result of the optimization study: 1550.18 Hz.

Reference

1. http://claymore.engineer.gvsu.edu/~schmitte/assign5.html.

Model Library path: `Structural_Mechanics_Module/Automotive_Applications/spinning_gear`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **2D** from the **Space dimension** list.

2 Select **Structural Mechanics Module>Plane Stress** in the list of application modes (see Figure 3-25).

3 Click **OK**.

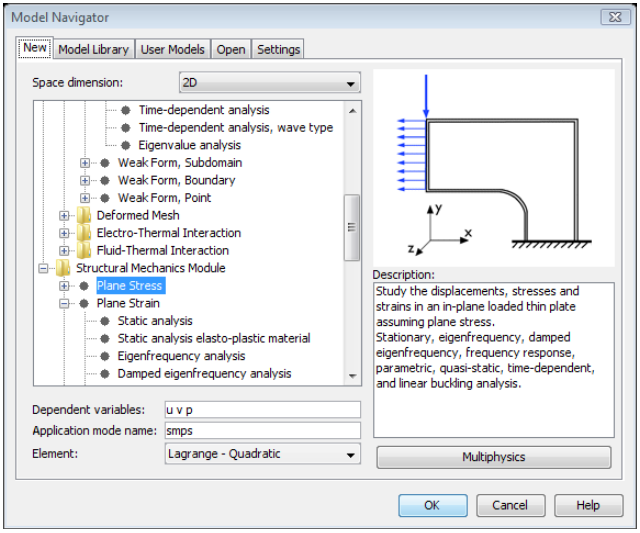


Figure 3-25: Selection of the Plane Stress application mode.

GEOMETRY MODELING

1 Choose **File>Import>CAD Data From File**.

2 Browse to `gear.dxf` in the `models/Structural_Mechanics_Module/Automotive_Applications` directory located in the COMSOL installation directory.

3 Click **Import**.

4 Make sure the geometry object is selected, then click the **Coerce to Solid** button on the Draw toolbar.

The imported geometry is not centered at the origin. However, centering the gear simplifies the modeling process.

Note: Selected geometry objects are highlighted in red and are subject to subsequent geometry operations. Ensure that the gear is highlighted before proceeding.

---

1 Click the **Move** button in the **Draw** toolbar.

2 Type `-1.228991` in the **x** edit field and `0.001573` in the **y** edit field.

3 Click **OK**. The gear is now centered at the origin.

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

Next, reduce the size of the imported object by a factor of 10:

5 Click the **Scale** button on the Draw toolbar. In the **Scale factor** area type `0.1` in both the **x** and the **y** edit field, then click **OK**.

6 Click **Zoom Extents**.

7 Click the **Rectangle/Square** button. Draw a square with opposite corner points at (0, 0) and (0.06, 0.06) by pointing at one corner and dragging to the opposite one using the right mouse button.

8 Copy the resulting object, SQ1, to the clipboard by pressing Ctrl+C. You will reuse this object in a later step.

Note: Using the right mouse button in Step 7 constrains the side lengths to be equal, thus resulting in the square object, SQ1. If you had used the left mouse button instead, the side lengths would not be constrained in this way and the created object would be a rectangle—therefore labeled R1—even though its sides were equal. In that case, just replace R1 for SQ1 in the following instructions.

---

9 Click the **Create Composite Object** button on the Draw toolbar.

10 Type `CO1*SQ1` in the **Set formula** edit field to do a Boolean intersection of the square and the gear, then click **OK**.

11 Shift-click the **Circle/Ellipse (Centered)** button in the Draw toolbar.

12 Type `0.015` in the **Radius** edit field, then click **OK**.

13 Press Ctrl+V to paste the stored object from the clipboard. Click **OK** to accept zero displacements from the original object.

14 Click the **Create Composite Object** button on the Draw toolbar.

15 Type `C1*SQ1` in the **Set formula** edit field to do a Boolean intersection of the square and the circle, then click **OK**.

16 Click the **Zoom Extents** button.

This completes the geometry. Compare your result with Figure 3-26.

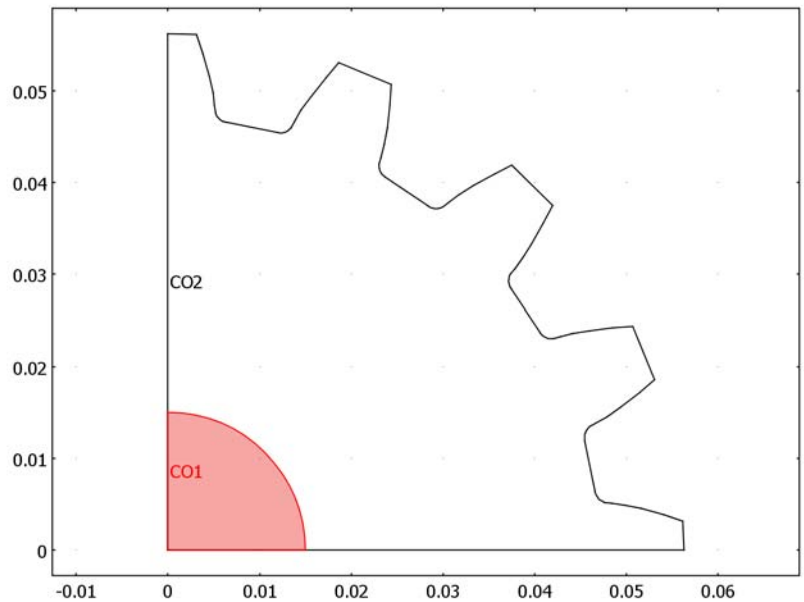


Figure 3-26: Completed shaft and gear geometries in 2D.

PHYSICS SETTINGS—INTERFERENCE FIT

Boundary Conditions

1 On the **Options** menu point to **Labels**, then click **Show Edge Labels**.

2 From the **Physics** menu choose **Boundary Settings**.

3 Select Boundaries 1, 2, 3, and 12 and select **Symmetry plane** from the **Constraint condition** list on the **Constraint** page.

4 Click **OK**.

Subdomain Settings

1 From the **Physics** menu choose **Subdomain Settings**.

2 Select Subdomains 1 and 2.

3 Click the **Load** button on the **Material** page to open the **Materials/Coefficients Library** dialog box.

4 Select the **Steel AISI 4340** material from the list of materials. Click **OK**.

5 Select Subdomain 1 (the shaft) and type `0.1` in the **thickness** edit field.

6 Select Subdomain 2 (the gear) and type `0.01` in the **thickness** edit field.

7 Click the **Load** tab and select Subdomain 2 only.

8 Select the **Include thermal expansion** check box.

9 Type `23[degC]` in the **Temp** edit field and type `700[degC]` in the **Tempref** edit field to set the strain temperature to 23 °C and the strain reference temperature to 700 °C. Click **OK**.

MESH GENERATION

1 Click the **Initialize Mesh** button in the Main toolbar.

The initial mesh is quite coarse at the interface between the shaft and the gear. Because the hoop stresses of the thermal interference drop steeply from this interface into the periphery of the gear, you need to refine the mesh at the interface.

2 From the **Mesh** menu choose **Free Mesh Parameters**.

3 Click the **Boundary** tab.

4 Select Boundary 45 in the geometry and type `1e-4` in the **Maximum element size** edit field.

5 Click **Remesh** and then click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to start the analysis.

POSTPROCESSING AND VISUALIZATION

1 From the **Postprocessing** menu, choose **Plot Parameters**.

2 Select the **Surface**, **Contour**, **Arrow**, **Max/min marker**, and **Deformed shape** check boxes on the **General** page to create a plot with these plot types.

3 Click the **Arrow** tab.

4 Select **Boundaries** in the **Plot arrows on** list. Click **OK**.

Compare your result with Figure 3-27.

1 For a second plot, click the **3D Surface Plot** button in the Plot toolbar.

2 From the **Postprocessing** menu, choose **Plot Parameters**.

3 Click the **General** tab.

4 Clear the **Deformed shape** check box and select the **Contour** and **Arrow** check boxes.

5 Click **OK**.

6 Click the **Headlight** button in the Camera toolbar.

You can rotate the plot by moving the mouse. Compare your result with Figure 3-28.

Note: There are spikes in the von Mises stress field which are due to singularities. They are caused by unsmooth corners in the imported CAD geometry.

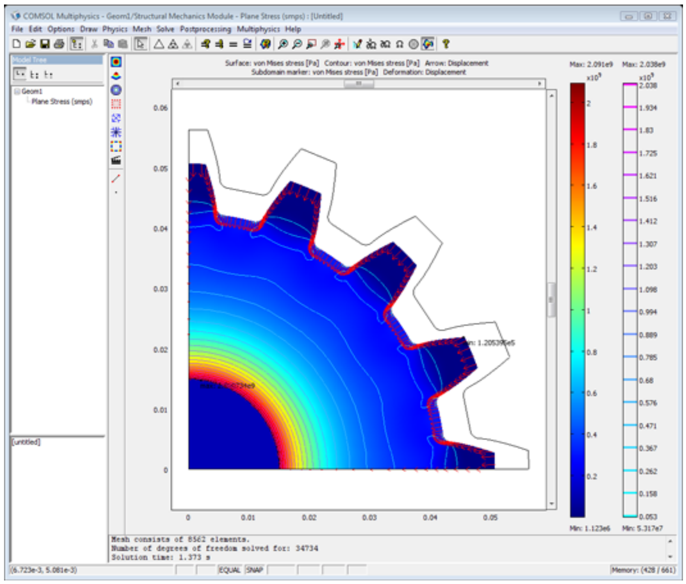


Figure 3-27: Von Mises stresses and deflection after thermal interference process.

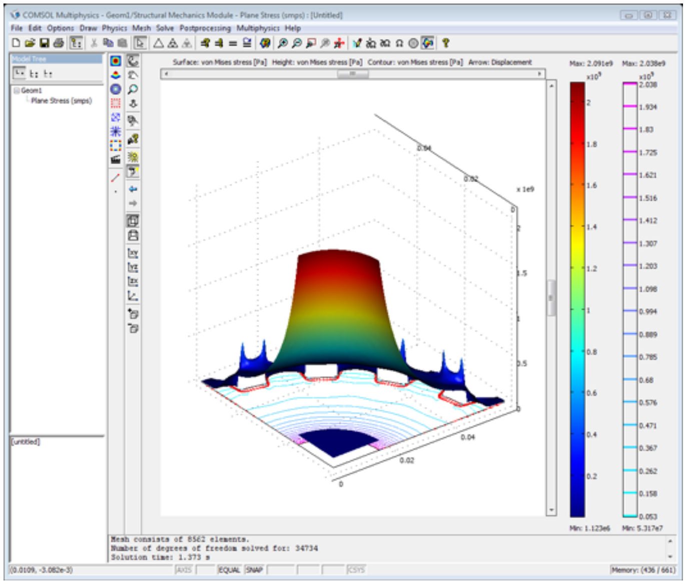


Figure 3-28: 3D surface plot of Figure 3-27.

This completes the first modeling stage.

SPINNING THE SHAFT-GEAR ASSEMBLY—SEPARATION FREQUENCY

The next analysis determines the separation frequency by spinning the shaft-gear assembly. In a strict sense, this is a contact problem and the description of the shaft-gear interface complex. However, because the resulting centripetal body load is nearly radially symmetric, it is not much of a liberty to assume that the entire contact interface separates at one specific frequency. Two body loads describe this problem:

- Residual stress field from the interference fit
- Rotational body force from spinning the assembly

To reference the interference stress fields and allow component mobility, add two additional Plane Stress application modes to the model and assign them to the shaft and gear domains, respectively. This enables a displacement comparison between the surfaces of the outer shaft and the inner gear. The independence of these two application modes implies that the displacement comparison has practical meaning only at frequencies greater than or equal to the separation frequency.

MULTIPHYSICS SETTINGS

1 From the **Multiphysics** menu choose **Model Navigator**.
2 With **Structural Mechanics Module>Plane Stress** already selected, click the **Add** button twice to add two more Plane Stress application modes: **Plane Stress (smps2)** and **Plane Stress (smps3)**.
3 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Constants**.
2 Enter the following constants in the **Name**, **Expression**, and **Description** (optional) columns; when done, click **OK**,

NAME	EXPRESSION	DESCRIPTION
shaftLength	0.1[m]	Shaft length
gearWidth	0.01[m]	Gear width
rho	7850[kg/m^3]	Density
f	1000[Hz]	Frequency

3 Click **OK**.
4 From the **Options** menu, point to **Expressions**, and then click **Scalar Expressions**.
5 Specify expressions according to the following table (the descriptions are optional):

NAME	EXPRESSION	DESCRIPTION
theta	atan2(y,x)	Angle
Force	4*pi^2*f^2*rho*sqrt(x^2+y^2)	Body load
Fx	Force*cos(theta)	Body load x-component
Fy	Force*sin(theta)	Body load y-component

6 Click **OK**.

When specifying expressions for the first time, it is useful to graph them by using postprocessing functions. To superimpose a vector plot of the body force onto the geometry, do as follows:

1 From the **Solve** menu, choose **Update Model** (this updates all expressions and makes them available for postprocessing).
2 From the **Postprocessing** menu, choose **Plot Parameters**.
3 Select the **Arrow** and **Geometry edges** check boxes only.
4 Click the **Arrow** tab.

5 Select **Subdomains** in the **Plot arrows on** list.

6 Type `Fx` and `Fy` in the **x component** and **y component** edit fields, respectively.

7 Click **OK** to display the body load **F**. Compare the result with Figure 3-30.

Figure 3-29 shows these plot settings for this plot.

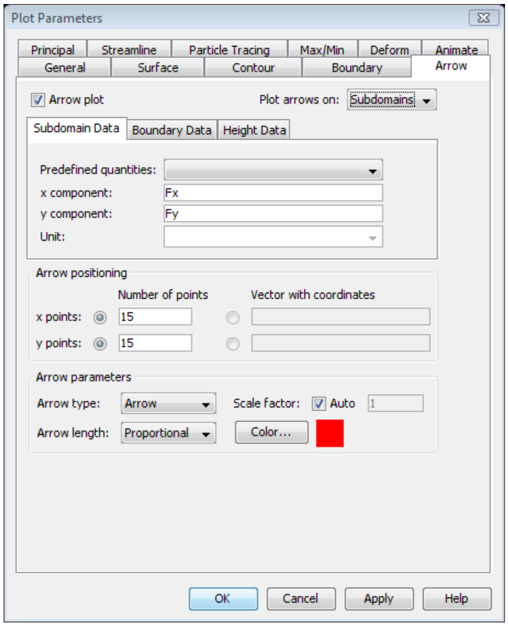


Figure 3-29: Plot settings for checking the centripetal body load.

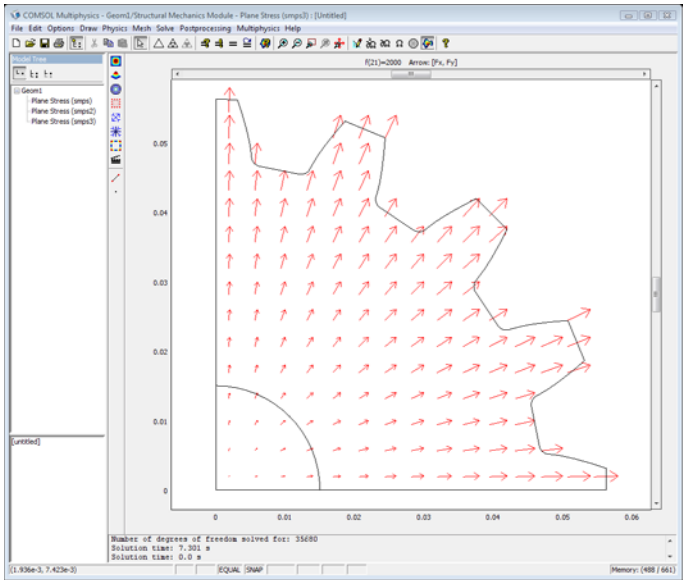


Figure 3-30: Graphical output based on input specified in Figure 3-29.

PHYSICS SETTINGS

Subdomain Settings—Plane Stress (smps3), Gear

Because it was created last, the application mode Plane Stress (smps3) is now active. This is indicated in the title bar of the COMSOL Multiphysics user interface and by a bullet in the **Multiphysics** menu. Assign the shaft to the Plane Stress (smps2) application mode and the gear to the Plane Stress (smps3) application mode by deactivating the other subdomain in the respective application modes:

1. From the **Physics** menu choose **Subdomain Settings**.
2. Select Subdomain 1 and clear the **Active in this domain** check box.
3. Select Subdomain 2 and click the **Material** tab.
4. Click the **Load** button to open the **Materials/Coefficients Library** dialog box.
5. Select the **Steel AISI 4340** material from the list of materials. Click **OK**.
6. Type the constant `gearWidth` in the **thickness** edit field.

7 Click the **Load** tab and specify the body load as follows:

QUANTITY	NAME	VALUE / EXPRESSION	UNITS
Body load x dir	F_x	Fx	N/m^3
Body load y dir	F_y	Fy	N/m^3

8 Click the **Body load is defined as force/volume using the thickness** option button.

9 Click the **Initial Stress and Strain** tab.

10 Select the **Include initial stress** check box and type the residual stresses `sx_smps`, `sy_smps`, and `sxy_smps` from the Plane Stress (smps) application mode in the edit fields for the initial normal stress and initial shear stress according to Figure 3-31.

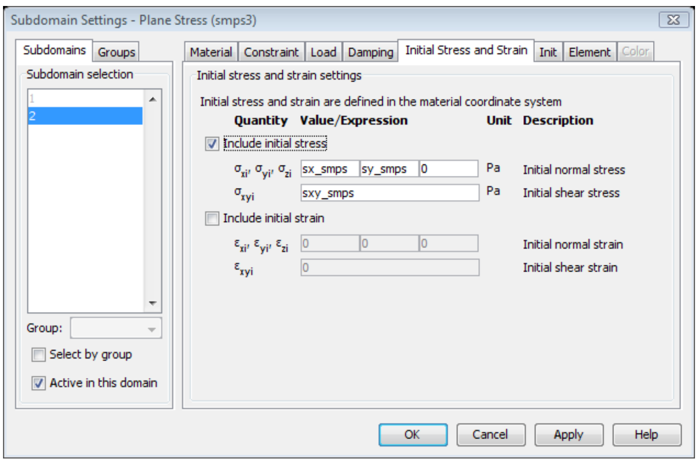


Figure 3-31: Specification of residual stresses sx_smps, sy_smps, and sxy_smps.

11 Click **OK**.

Subdomain Settings—Plane Stress (smps2), Shaft

1 From the **Multiphysics** menu, choose **Plane Stress (smps2)**.

2 From the **Physics** menu, choose **Subdomain Settings**.

3 Clear the **Active in this domain** check box for the already selected Subdomain 2.

4 Select Subdomain 1.

5 Click the **Load** button on the **Material** page to open the **Materials/Coefficients Library** dialog box.

6 Select the **Steel AISI 4340** material from the list of materials. Click **OK**.

7 Type the constant `shaftLength` in the **thickness** edit field.

8 Specify the same load and initial stress and strain expressions as for the Plane Stress (smps3) application mode.

9 Click **OK** to exit the **Subdomain Settings** dialog box.

Boundary Conditions

1 From the **Physics** menu, choose **Boundary Settings**.

2 Select Boundaries 1 and 2 and select **Symmetry plane** in the **Constraint condition** list.

3 Click **OK**.

4 From the **Multiphysics** menu, choose **Plane Stress (smps3)**.

5 From the **Physics** menu, choose **Boundary Settings**.

6 Select Boundaries 3 and 12 and select **Symmetry plane** in the **Constraint condition** list.

7 Click **OK**.

Options and Settings—Integration Coupling Variable

1 From the **Options** menu, point to **Integration Coupling Variables**, and then click **Boundary Variables**.

2 Select Boundary 45 and define the integration coupling variable `int_disp` according to the following table:

QUANTITY	NAME	EXPRESSION
Displacement integral	`int_disp`	`((u3-u2)^2+(v3-v2)^2)^0.5`

3 Click **OK**.

COMPUTING THE SOLUTION

The residual stresses `sx_smps`, `sy_smps`, and `sxy_smps` and the added centripetal body loads are now used to compute the relative displacements of the gear and the shaft. Because the residual stresses are already available from the previous computation, the next step instructs COMSOL Multiphysics to use this existing data. There is no need to recompute the solution for the Plane Stress (smps) application mode:

1 Click the **Solver Manager** button and click the **Current solution** option button in the **Values of variables not solved for and linearization point** area.

2 Click the **Solve For** tab.

3 Select **Plane Stress (smps2)** and **Plane Stress (smps3)** such that both of them are highlighted. (Ctrl-clicking accomplishes this.)

4 Click **OK**.

At this point, it is unknown at which frequency the shaft and the gear separate. More time effective than many trial and error iterations of single test frequencies is a frequency sweep. For instance, specify a sweep from 1000 Hz to 2000 Hz in increments of 50 Hz as follows:

1. Click the **Solver Parameters** button.
2. Select **Parametric** from the **Solver** list.
3. On the **General** page, type `f` in the **Parameter name** edit field and the vector `1000:50:2000` in the **Parameter values** edit field.
4. Click **OK**.
5. Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1. From the **Postprocessing** menu choose **Cross-Section Plot Parameters**.
2. In the **Plot type** area on the **General** page click the **Point plot** option button.
3. All parametric solutions in the **Solutions to use** list are highlighted by default.
4. Click the **Point** tab.
5. Type `int_disp` in the **Expression** edit field.
6. Click **OK**.

Figure 3-32 reveals that the gear separates from the shaft at approximately 1550 Hz.

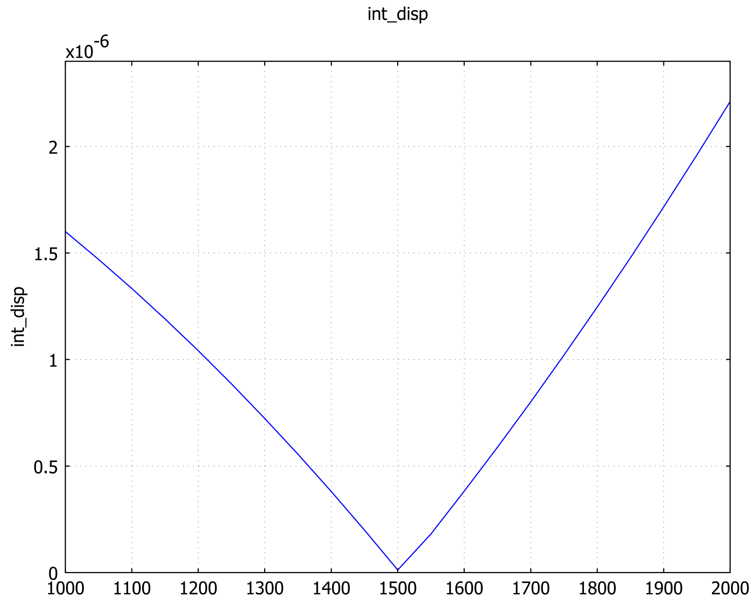


Figure 3-32: Displacement versus frequency; separation occurs at 1550 Hz.

To plot the von Mises stress field at 1600 Hz, follow this procedure:

1 Close Figure 3-32.
2 On the **Options** menu, point to **Expressions**, and then click **Subdomain Expressions**.
3 Select Subdomain 1.
4 Define the variables `mises_all`, `u_all`, and `v_all`, and specify the corresponding expressions `mises_smps2`, `u2`, and `v2`, respectively.
5 Select Subdomain 2 and specify the expressions `mises_smps3`, `u3`, and `v3`.
6 Click **OK**.
7 To update these newly introduced subdomain expressions with the current solution, choose **Update Model** from the **Solve** menu.
8 From the **Postprocessing** menu choose **Plot Parameters**.
9 On the **General** page select the **Surface**, **Deformed shape**, and **Geometry edges** check boxes only.
10 In the **Solution to use** area select `1600` in the **Parameter value** list.

11 Click the **Surface** tab. On the **Surface Data** page type `mises_all` in the **Expression** edit field. On the **Height Data** page clear the **Height data** check box.

12 Click the **Deform** tab, then click the **Subdomain Data** tab in the **Deformation data** area. In the **x component** edit field type `u_all` and in the **y component** edit field type `v_all`.

13 Click **OK**.

14 Click the **Zoom Extents** button.

Note: At 1600 Hz, the assembly has passed the separation point and a gap between the components is visible in Figure 3-33.

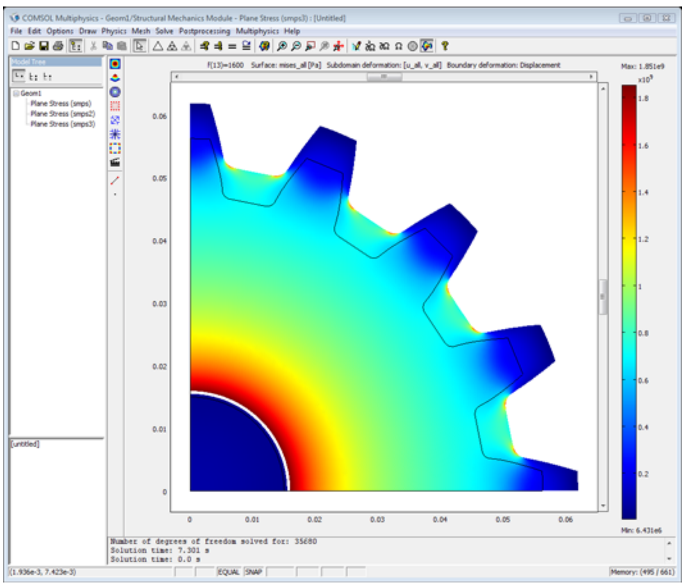


Figure 3-33: Von Mises stresses and deflection at 1600 Hz.

Minimizing the Model and Determining the Separation Frequency

---

Note: This section requires that you run COMSOL Script or MATLAB.

Finding the separation frequency is an inverse problem. The parametric solver enables you to find increasingly accurate solutions via narrowing the sweep range by inspection. A more systematic solution uses M-file scripting in which you can obtain a result within a specified tolerance by programmatic iterations.

The resulting physics model describes the spinning gear at any frequency. The optimization problem is the inverse: find the frequency at which the gear and shaft separate. The optimization equation is

$$\underset{x}{\text{minimize}} \qquad \mathrm{intdisp}(x)$$

where intdisp is the displacement integral.

The optimization routine does not need to know the details of the physics model, it simply queries for the distance between shaft and gear at any given frequency.

OPTIMIZING ON THE COMMAND LINE

Export the model to COMSOL Script by choosing **File>Export>FEM structure as 'fem'**. COMSOL Script opens automatically.

The spinning gear model should now be present in the workspace as the variable `fem`.

The following functions solve the optimization problem:

```
[freq,int_disp] = cl_gear(fem,arclength)
f = cl_gear_obj(x,fem,arclength)
fem = cl_gear_minimal(starting_point)
```

Note: All these functions are available from COMSOL Script or MATLAB.

`cl_gear_obj` defines the value of the objective at a given frequency `x`:

```
% Solve the problem at frequency x
fem.sol=femlin(fem, ...
               'solcomp',{'u2','v3','v2','u3'}, ...
               'outcomp',{'u2','v3','u','v2','u3','v'}, ...
               'pname','f', ...
               'plist',x, ...
               'nonlin','off');
% Pick up displacement integral
pd = posteval(fem,'int_disp','edim',0,'dl',1);
% Normalize
f = pd.d/arclength;
```

`cl_gear` is the main function that defines the optimization problem and calls the solver.

```
% Start from first existing solution
% (e.g. opt.init.x = 1000;)
x0 = fem.sol.plist(1);
% Solve
[freq,int_disp] = fminsearch('cl_gear_obj',x0,[],fem,arclength);
```

For the initial solution, this script uses the first solution among those in the physics model although it is not the best one. This is for demonstration purposes only; it is always advisable to start with the best available solution. (In this case, the initial sweep range happens to have hit very close to the optimum at a frequency of 1550 Hz; this would normally be the solution from which to begin optimization.)

Note that `cl_gear` also outputs three plots: the separation frequency during optimization (Figure 3-34), and the stress distribution at separation as a 2D (Figure 3-35) and 3D plot (Figure 3-36).

To run the optimization call the `cl_gear` function:

```
[freq,int_disp]=cl_gear(fem,(2*pi*0.015)/4)
```

This prints the frequency and distance during the iterations and returns an optimal solution of 1550.19 Hz. (The second argument, `(2*pi*0.015)/4`, is the length of the circle segment and is used in `cl_gear_obj` to normalize the displacement integral.)

You can also solve this problem using the solvers of the COMSOL Optimization Lab. See “Spinning Gear” in Chapter 4 of the *Optimization Lab User’s Guide* for details.

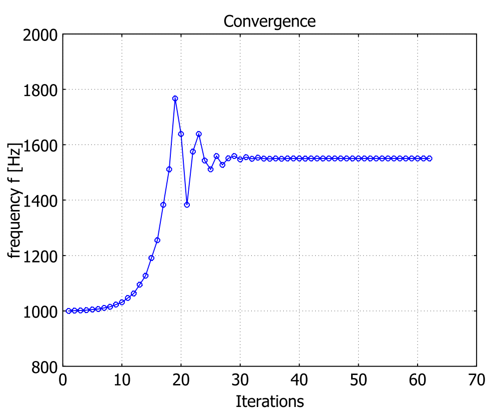


Figure 3-34: Convergence of the separation frequency during optimization.

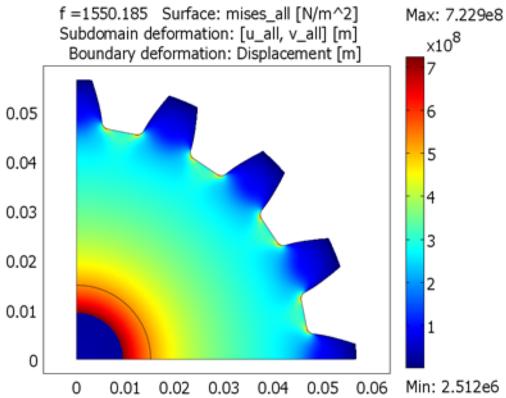


Figure 3-35: Stress distribution at separation.

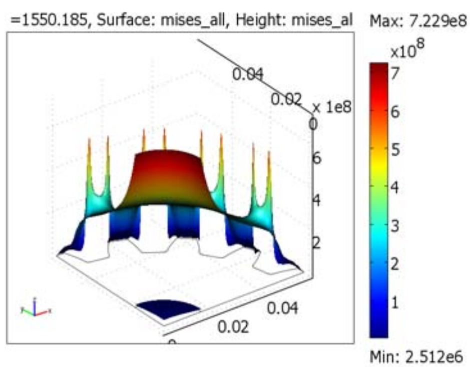


Figure 3-36: Stress distribution at separation.

EXPLOITING MODEL SYMMETRY

To reduce the required computation time, you can exploit the remaining model symmetries instead of solving the complete model as above. Figure 3-37 illustrates the stresses superimposed on the minimal model at the separation frequency.

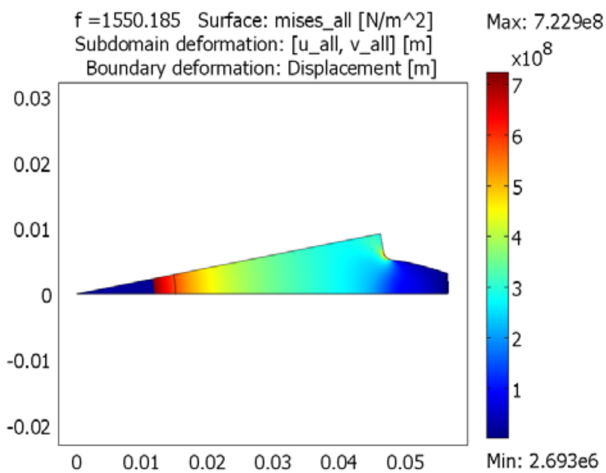


Figure 3-37: Von Mises stresses superimposed on the minimal model

To run the example on the minimal model, load the modified FEM structure, then proceed as before.

1 At the prompt, type:

```
fem = cl_gear_minimal(1000);
```

This solves the reduced physics problem at an initial frequency of 1000 Hz.

2 Solve the optimization problem as before, but normalize with the reduced segment length (1/32 of the full circle). Type:

```
[freq,int_disp]=cl_gear(fem,(2*pi*0.015)/32);
```

Figure 3-38 shows the resulting stress distribution plot.

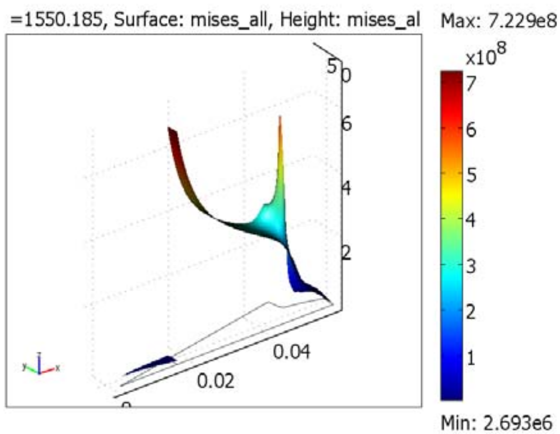


Figure 3-38: Stress distribution at separation.

Automobile Wheel Rim

Introduction

In this model you analyze the stress distribution in a lightweight automobile wheel rim. Because the wheel is rotating, the analysis includes a number of different load cases. By using symmetric and antisymmetric boundary conditions together with superposition of load cases, you can perform the modeling in a fast and efficient way.

Model Definition

The wheel rim for this analysis is a ten-spoke model where the design elements of the geometry cause the finite element mesh to become quite large. To reduce the size of the problem, you can use an iterative solver and make use of the symmetry. Loading on the tire is composed of both the tire pressure and a rotating load transferred from the tire to the rim. In this case, only the geometry is symmetric, while the moving load is not. Because the problem is linear, you can use superposition of load cases. Any load on a symmetric structure can be separated into one symmetric and one antisymmetric load, as illustrated in Figure 3-39.

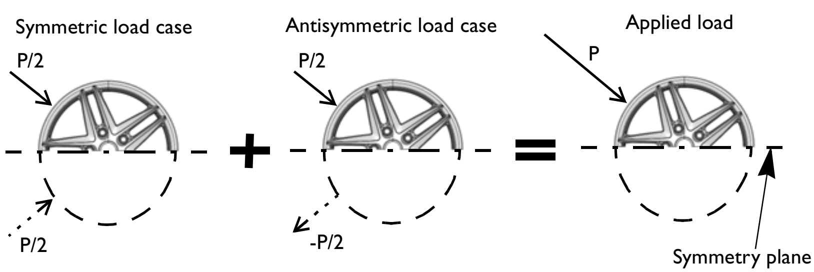


Figure 3-39: Superposition of a symmetric and an antisymmetric load case results in the total applied load to the symmetric geometry of the wheel rim.

For an applied load at a certain position, you can solve the model using one half of the geometry with symmetric and antisymmetric boundary conditions, respectively. As illustrated above, the two superposed solutions (symmetric + antisymmetric) represent the total solution.

In the analysis, you study the problem in a coordinate system fixed to the wheel rim, where the load rotates around the wheel. Assume that the load on the rim extends 30° in each direction from the point of contact. It seems reasonable to analyze the stress distribution in the rim as the load passes by one of the spokes. For the current geometry, this movement is equivalent to a 36° rotation of the point of contact, which you can choose to divide into five load positions with 9° intervals between them. In the following figure you can see the studied locations, given by an angle of rotation in the rim's coordinate system.

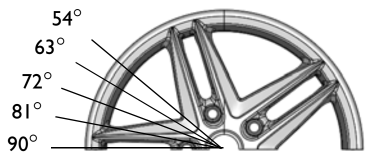


Figure 3-40: Angular positions for the point of contact for the chosen locations.

You can model the moving load by letting the parametric solver step through the values of a parameter, which you can then use in the load expression to control its location and distribution. Use Table 3-1 and Table 3-2 to find each load case by its parameter value and load angle. Combine the load cases according to Table 3-3 to calculate the stress distribution for load positions around the entire wheel. The designations used in the Combination column correspond to load cases listed in Table 3-1 and Table 3-2.

TABLE 3-1: SYMMETRIC LOAD CASES

PARAMETER VALUE	DESCRIPTION	DESIGNATION
1	Rotating load 90°	sr1
2	Rotating load 81°	sr2
3	Rotating load 72°	sr3
4	Rotating load 63°	sr4
5	Rotating load 54°	sr5
6	Tire pressure	st6

TABLE 3-2: ANTISYMMETRIC LOAD CASES

PARAMETER VALUE	DESCRIPTION	DESIGNATION
2	Rotating load 81°	ar2
3	Rotating load 72°	ar3
4	Rotating load 63°	ar4
5	Rotating load 54°	ar5

TABLE 3-3: COMBINED LOAD CASES

PARAMETER VALUE	DESCRIPTION	COMBINATION
0	Tire pressure + Rotating load 90°	st6 + sr1
9	Tire pressure + Rotating load 81°	st6 + (sr2 + ar2)/2
18	Tire pressure + Rotating load 72°	st6 + (sr3 + ar3)/2
27	Tire pressure + Rotating load 63°	st6 + (sr4 + ar4)/2
36	Tire pressure + Rotating load 54°	st6 + (sr5 + ar5)/2

MATERIAL

Assume that the wheel rim is made of aluminum which has a Young's modulus E = 70 GPa and a Poisson's ratio ν = 0.33.

CONSTRAINTS

- A region around each bolt hole is fixed.

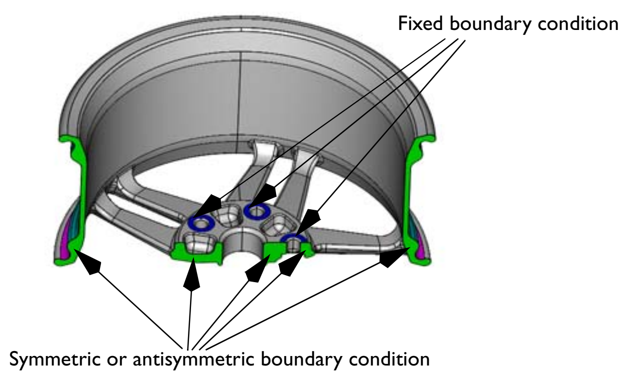


Figure 3-41: Constrained boundaries of the wheel rim.

- Symmetric load cases: Symmetry condition (normal displacement fixed) in the symmetry plane.
- Antisymmetric load cases: Antisymmetry condition (transverse displacement fixed) in the symmetry plane.

LOADS

- Tire pressure: The overpressure is 2 bar = 200 kPa. Use the last parameter step to apply this load.
- Rotating ("road") load: The total load carried by the wheel corresponds to a weight of 1120 kg. Apply this as a pressure on the rim surfaces where the tire is in contact. Assume that the load distribution in the circumferential direction can be approximated as $p = p_0 \cos(3\vartheta)$ where ϑ is the angle from the point of contact between the rim and the tire. The loaded area thus extends 30° in each direction. The solver parameter, k_{solv}, in the following expression is used to control the location of the load:

$$p = -p_0(k_{\text{solv}} < 6)\cos\left[3\,\text{atan}\left(\frac{y}{x}\right) - \frac{3\pi(11 - k_{\text{solv}})}{20}\right] \cdot$$

$$\cdot \left[\text{atan}\left(\frac{y}{x}\right) < \frac{\pi(43 - 3k_{\text{solv}})}{60}\right] \cdot \left[\text{atan}\left(\frac{y}{x}\right) > \frac{\pi(23 - 3k_{\text{solv}})}{60}\right]$$

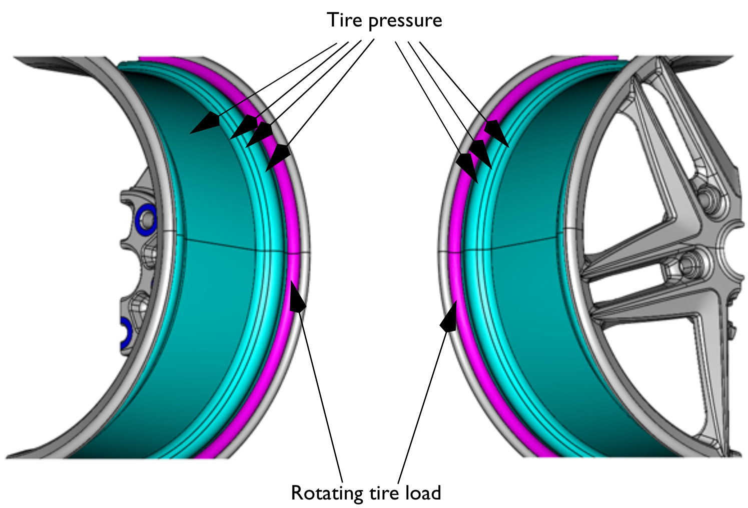


Figure 3-42: Applied loads.

THE FINITE ELEMENT MESH

Meshing of curved surfaces, like many of those found in the geometry of the wheel rim, can cause inverted surface elements. By default these are automatically linearized to avoid problems during solution. The linearized elements can however give rise to high local stresses, since they can introduce sharp corners into the surface mesh. This does not influence the global accuracy of the solution, but you may be required to manually set the range for the stress diagrams during postprocessing due to high local stresses. For the regions around the spoke, you can refine the surface mesh, to avoid inverted and thus linearized elements.

The final finite element mesh consists of approximately 126,000 tetrahedral elements, giving a total of approximately 615,000 degrees of freedom.

Results

The figure below shows the von Mises stress for the rotating load at 90° superposed with the tire pressure. The region with the highest stresses is at the foot of the spoke where the load is located.

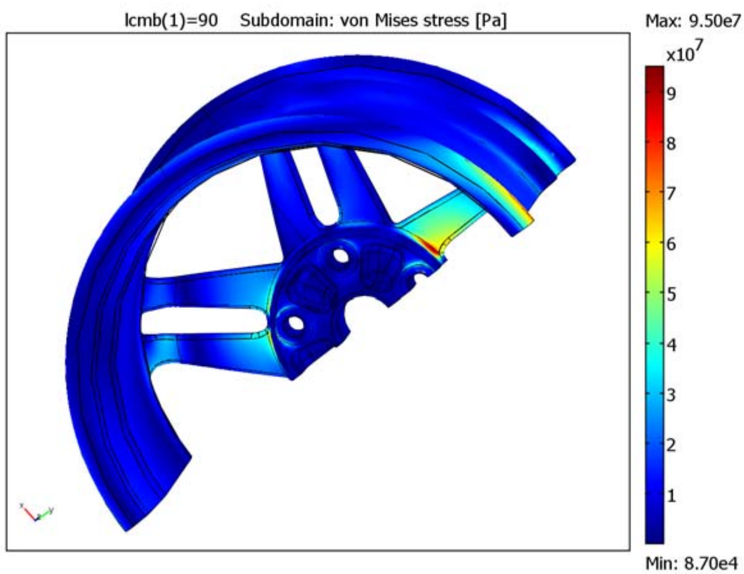


Figure 3-43: The von Mises stress distribution in the wheel rim under the combined loading of the tire pressure and the rotating load at the 90° position.

The von Mises stress for the remaining point of contact positions, 81°, 72°, 63°, and 54°, superposed with the inner pressure of the tire are shown below, illustrating how the stress field moves along the load path.

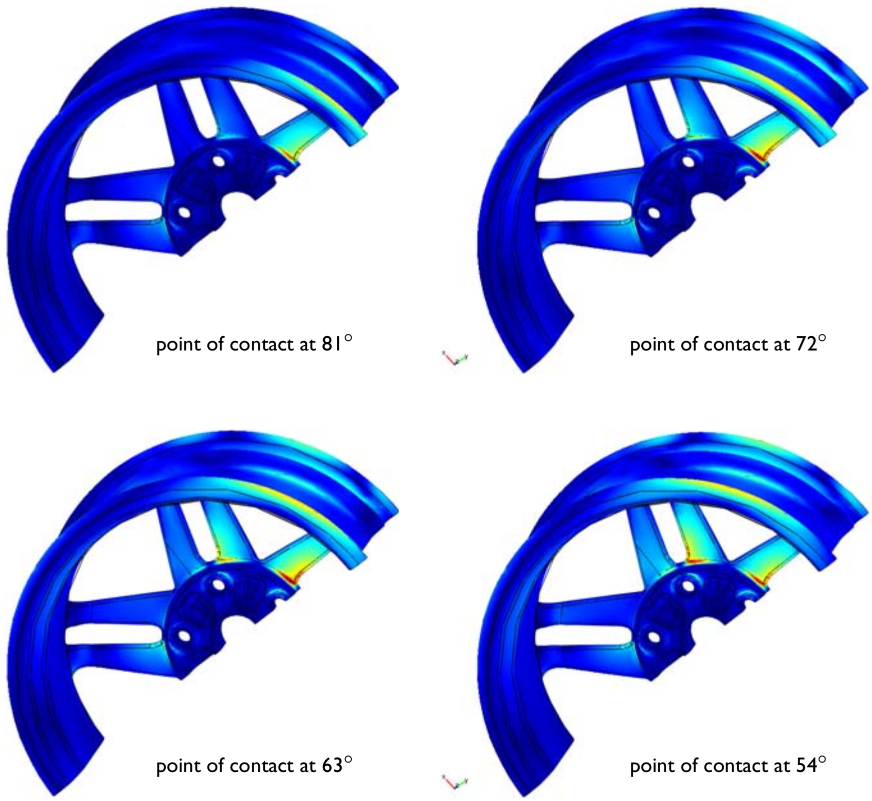


Figure 3-44: The von Mises stress distribution for the combined loading of the tire pressure and the rotating load at the locations specified in the figure. The color scale is the same as in Figure 3-43.

Modeling in COMSOL Multiphysics

You can build and solve two models, one containing the symmetric load cases and the pressure load and a second one containing the antisymmetric load cases.

To superpose the symmetric and antisymmetric load cases, export the FEM structures from these two models to COMSOL Script where you can extract and add the solution vectors.

Finally, to visualize the solution, import the FEM structure with the superposed solution back into the COMSOL Multiphysics user interface.

Model Library path: `Structural_Mechanics_Module/Automotive_Applications/wheel_rim`

Modeling in the Graphical User Interface

MODEL NAVIGATOR

1. Click the **New** button or start COMSOL Multiphysics to open the **Model Navigator**.
2. Select **3D** from the **Space dimension** list.
3. Open the **Structural Mechanics Module** folder and then **Solid, Stress-Strain**. Select **Parametric analysis**.
4. Click **OK**.

IMPORT OF CAD GEOMETRY

1. On the **File** menu select **Import>CAD Data From File**.
2. In the **Files of type** list select **COMSOL Multiphysics file**.
3. Browse to the `models/Structural_Mechanics_Module/Automotive_Applications` directory located in the COMSOL installation directory and select the file `wheel_rim.mphbin`.
4. Click **Import**.

OPTIONS AND SETTINGS

1. Select **Options>Constants**.
2. Enter the following constants in the **Constants** dialog box; when done, click **OK**.

NAME	EXPRESSION
`pressure`	`2e5[Pa]`
`tire_load`	`5.194e6[Pa]`

3 Select **Options>Visualization/Selection Settings**.

4 On the **Rendering/Selection** page, select **Coarse** from the **Visualization mesh** list.

This setting reduces the time it takes to render the geometry when switching modes in the user interface.

5 Click **OK**.

MESH GENERATION

1 Select **Mesh>Free Mesh Parameters**.

2 On the **Boundary** page select boundaries and enter the parameters according to the following table.

BOUNDARIES	MESH CURVATURE FACTOR	MESH CURVATURE CUTOFF
89, 90	0.5	
72, 85, 94, 96, 98, 106, 107, 132, 197, 206, 265	0.4	
84, 92, 95, 131, 229, 251	0.4	0.02
74, 97, 145, 148, 177, 196, 207, 228	0.4	0.015

3 Click **OK**.

4 Click the **Mesh All (Free)** button in the Mesh toolbar.

PHYSICS SETTINGS (SYMMETRIC LOAD CASES)

Subdomain Settings

1 On the **Physics** menu select **Subdomain Settings**.

2 In the **Subdomain selection** list select subdomain 1.

3 On the **Material** page click the **Load** button to open the **Materials/Coefficients Library** dialog box.

4 In the **Materials** list expand **Basic Material Properties** then select **Aluminum**.

5 Click **OK** to apply the material and close the dialog box.

6 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

1 From the **Physics** menu choose **Boundary Settings**.

2 On the **Boundaries** page locate the **Boundary selection** list and select Boundaries 56, 156, and 232.

3 On the **Constraint** page select **Fixed** from the **Constraint condition** list.

4 In the **Boundary selection** list select Boundaries 1, 26, 48, 64, and 76.

5 On the **Constraint** page select **Symmetry plane** from the **Constraint condition** list.

6 In the **Boundary selection** list select Boundaries 10–15, 18, 19, 22, 81–83, 86, 87, 89–91, and 93.

7 On the **Load** page select **Distributed load** from the **Type of load** list.

8 Select **Tangent and normal coord. sys. (t_1, t_2, n)** from the **Coordinate system** list.

9 Enter `-pressure*(load_param>5)` in the **F_n** edit field.

10 In the **Boundary selection** list select Boundaries 6, 7, 92 and 94.

11 On the **Load** page select **Distributed load** from the **Type of load** list.

12 Select **Tangent and normal coord. sys. (t_1, t_2, n)** from the **Coordinate system** list.

13 In the **F_n** edit field enter `-tire_load*(load_param<6)*cos(3*atan2(y,x)-3*pi/20*(11-load_param))*(atan2(y,x)<(pi/60*(43-3*load_param)))*(atan2(y,x)>(pi/60*(23-3*load_param)))`

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

COMPUTING THE SOLUTION (SYMMETRIC LOAD CASES)

1 Click the **Solver Parameters** button on the Main toolbar.

2 On the **General** page, type `load_param` in the **Parameter name** edit field and the vector `1:6` in the **Parameter values** edit field.

3 In the **Linear system solver** list select **Conjugate gradients**.

4 In the **Preconditioner** list select **Geometric multigrid**.

5 Click the **Settings** button.

6 In the dialog box that opens select **Linear system solver>Preconditioner>Coarse solver**.

7 In the **Coarse solver** list box select **SPOOLES**.

8 Click **OK**.

9 Click **OK**.

10 Click the **Solve** button on the Main toolbar.

SAVING THE SYMMETRIC LOAD CASES

After the problem is solved you can optionally save the model, in case you want to open and examine the solution to the symmetric load cases. If you do not wish to save the model continue with the next section.

1 Select **File>Save**.
2 Navigate to the directory of your choice.
3 In the **File name** edit field enter `wheel_rim_sym.mph`.
4 Click **OK**.

A solved model with the symmetric load cases is available in the COMSOL installation directory under `models/Structural_Mechanics_Module/Automotive_Applications/wheel_rim_sym.mph`.

EXPORTING THE SYMMETRIC LOAD CASES TO COMSOL SCRIPT

You can export the model to COMSOL Script as an FEM structure containing the geometry, mesh, physics settings, as well as the solution. From the FEM structure you can then extract the solution vector, which you can add to the solution vector of the antisymmetric load cases.

1 Select **File>Export>FEM Structure**.
2 In the dialog box that opens enter `fem_sym_cases` in the **Variable name for FEM structure** edit field.
3 Click **OK**. COMSOL Script opens automatically.
4 Switch back to COMSOL Multiphysics before continuing with the next step.

PHYSICS SETTINGS (ANTISYMMETRIC LOAD CASES)

On the symmetry plane of the geometry you need to change the boundary condition to antisymmetry before computing the antisymmetric load cases. Also, you can skip the last parameter step because it corresponds to the pressure load.

Boundary Conditions

1 On the **Physics** menu select **Boundary Settings**.
2 In the **Boundary selection** list select Boundaries 1, 26, 48, 64, and 76.
3 On the **Constraint** page select **Antisymmetry plane** from the **Constraint condition** list.
4 Click **OK** to close the **Boundary Settings** dialog box.

COMPUTING THE SOLUTION (ANTISYMMETRIC LOAD CASES)

1 Click the **Solver Parameters** button on the Main toolbar.
2 On the **General** page, change the vector to `2:5` in the **Parameter values** edit field.
3 Click **OK**.
4 Click the **Solve** button on the Main toolbar.

SAVING THE ANTISYMMETRIC LOAD CASES

Carry out the following steps to optionally save the model at this stage. If you do not wish to save the model continue with the next section.

1 Select **File>Save As**.
2 Navigate to the directory of your choice.
3 In the **File name** edit field enter `wheel_rim_asym.mph`.
4 Click **Save**.

A solved model with the antisymmetric load cases is available in the COMSOL installation directory under `models/Structural_Mechanics_Module/Automotive_Applications/wheel_rim_asym.mph`.

EXPORTING THE ANTISYMMETRIC LOAD CASES TO COMSOL SCRIPT

After the antisymmetric model is solved you can export the solution to COMSOL Script.

1 Select **File>Export>FEM Structure**.
2 In the dialog box that opens enter `fem_asym_cases` in the **Variable name for FEM structure** edit field.
3 Click **OK**. COMSOL Script opens automatically.

POSTPROCESSING AND VISUALIZATION

To obtain the total solution you need to superpose, or add, the solutions from the pressure load and the symmetric and antisymmetric load cases. With the two FEM structures exported to COMSOL Script add the solution vectors according to Table 3-3 and create a new FEM structure, which you can import into the graphical user interface to visualize the results. To do this use the script `wheel_rim_superpose.m`, which you find in the `sme` directory located in the COMSOL installation directory.

1 With both FEM structures exported to COMSOL Script, run the script `wheel_rim_superpose.m` by typing `wheel_rim_superpose` at the prompt and pressing Return.
2 Switch back to COMSOL Multiphysics and select **File>Import>FEM Structure**.
3 In the **Enter name of FEM structure variable** enter `fem_added`.
4 Click **OK**.
5 Click the **Plot Parameters** button on the Main toolbar.
6 Select the **General** page.

7 In the **Plot type** area clear the **Slice** check box.

8 Select the **Subdomain** check box.

9 In the **Parameter value** list select **54** to display the von Mises stress for the point of contact at 54°.

10 Select the **Make rough plots** check box.

The above feature generates plots with a bit lower quality than normal for faster and more memory-efficient rendering.

11 Click **OK**.

4

Benchmark Models

This section contains examples of benchmarks models. You find more benchmark models in the *Structural Mechanics Module User's Guide*.

Wrapped Thick Cylinder Under Pressure and Thermal Loading

Introduction

This is a benchmark model for composite-materials analysis published by NAFEMS (Ref. 1). A detailed description of the problem as well as analytical results enable comparisons between the COMSOL Multiphysics results and the benchmark values.

This is also a modified version of a test used by Taig (Ref. 2).

Model Definition

The geometry is a long, thick, and hollow cylinder consisting of two layers. The inner layer, with an outer radius of 25 mm, is made of an isotropic material, while the outer layer (outer radius of 27 mm) is made of an orthotropic material. The length of the cylinder is 200 mm. The material properties of the outer layer are equal in the radial and axial directions, and differ from those in the angular (hoop) direction.

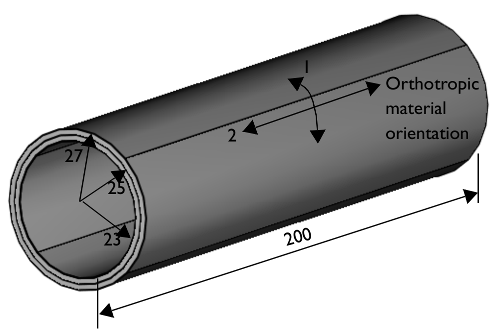


Figure 4-1: Geometry of the problem (all dimensions in mm).

Two load cases are considered:

1. An internal pressure of 200 MPa is applied.
2. An internal pressure of 200 MPa is applied in combination with a uniform increase in the temperature by 130 K.

The purpose of including thermal effects in the second case is to simulate prestress caused by tension in the hoop windings.

The material properties for the isotropic material are:

- E = 210 GPa, ν = 0.3, $\alpha = 2\cdot 10^{-5}\ \mathrm{K}^{-1}$

For the orthotropic material the properties read:

- E_1 = 130 GPa, $E_2 = E_3$ = 5 GPa
- $\nu_{12} = \nu_{13} = 0.25$, $\nu_{23} = 0$
- $G_{12} = G_{13}$ = 10 GPa, G_{23} = 5 GPa
- $\alpha_1 = 3\cdot 10^{-6}\ \mathrm{K}^{-1}$, $\alpha_2 = \alpha_3 = 2\cdot 10^{-5}\ \mathrm{K}^{-1}$

Because of the circular symmetry of the geometry, it is sufficient to model a quarter cylinder, with zero normal displacement imposed as the boundary condition on each of the two perpendicular rectangular faces. Reflection symmetry furthermore implies that half of the length of the cylinder can be left out by setting the displacement in the axial direction to zero on the annular cross section at the middle of the cylinder.

The target solution given by NAFEMS is:

TABLE 4-1: NAFEMS TARGET FOR THE TWO CASES.

POSITION	CASE 1	CASE2
Hoop stress at inner surface (isotropic)	1565.3 MPa	1381 MPa
Hoop stress at interface (isotropic)	1429.7 MPa	1259.6 MPa
Hoop stress at interface (orthotropic)	874.7 MPa	1056 MPa
Hoop stress at outer surface (orthotropic)	759.1 MPa	936.1 MPa

These results were obtained using standard equations for a compound cylinder, modified to take the orthotropic nature of the outer material into account. For a long cylinder, the so obtained solution is acceptable away from the ends. For this reason, Table 4-1 displays the solution evaluated at the annular cross section in the middle of the cylinder.

Modeling in COMSOL Multiphysics

For this model, you can take advantage of the possibility to implement user-defined coordinate systems in the Structural Mechanics Module by defining the material properties in cylindrical coordinates. The resulting relation between the material orientation and the coordinate labels in the user interface is as follows:

TABLE 4-2: RELATION BETWEEN MATERIAL ORIENTATION AND USER INTERFACE COORDINATE LABELS.

DESCRIPTION	MATERIAL ORIENTATION	USER INTERFACE COORDINATE LABEL
angular direction	1	y
axial direction	2	z
radial direction	3	x

Note that material properties sometimes are given in an orientation that differs from the one required by COMSOL Multiphysics. This is such a case, since the orthotropic material data are ν_{12}, ν_{13}, and ν_{23}, whereas COMSOL Multiphysics requires that you enter the Poisson ratios as ν_{xy}, ν_{yz}, and ν_{xz}—that is, ν_{31}, ν_{12}, and ν_{32}, respectively. Since the Poisson ratios are not symmetric, each parameter must be reevaluated in the correct orientation. The relation between the ratios for the orthotropic case reads:

$$\frac{\nu_{ij}}{E_i} = \frac{\nu_{ji}}{E_j}$$

Hence

$$\nu_{xy} = \frac{\nu_{yx} \cdot E_x}{E_y} = \frac{\nu_{13} \cdot E_3}{E_1}$$

and

$$\nu_{xz} = \frac{\nu_{zx} \cdot E_x}{E_z} = \frac{\nu_{23} \cdot E_3}{E_2} = 0$$

The shear moduli, G_{ij}, are symmetric, and you can therefore enter the given values for those directly in COMSOL Multiphysics.

Results and Discussion

A cross-section plot of the hoop stress—that is, the normal stress along the angular direction—gives the following result:

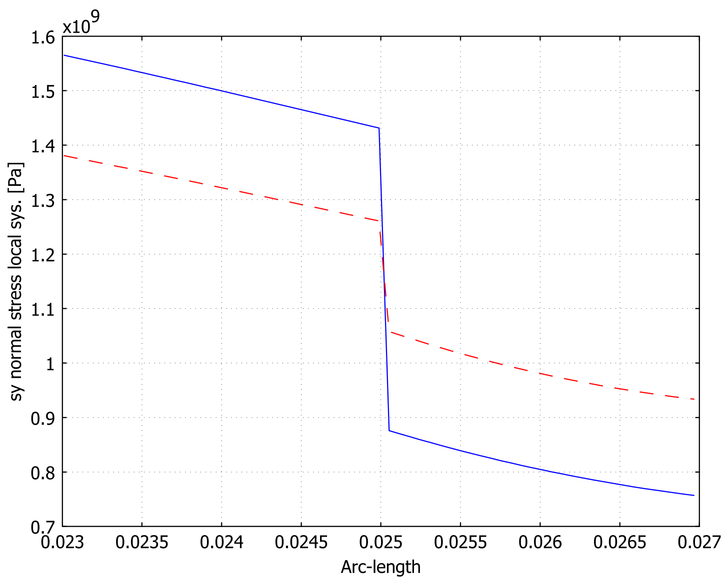


Figure 4-2: Cross-sectional plot of the hoop stress through the tube at z=0 (red dashed line: case2).

TABLE 4-3: COMPARISON OF THE COMSOL MULTIPHYSICS RESULTS VS. NAFEMS TARGET FOR CASE 1.

POSITION	**COMSOL MULTIPHYSICS RESULTS CASE1**	**NAFEMS TARGET** case1	**ERROR**
Hoop stress at inner surface (isotropic)	1568.5 MPa	1565.3 MPa	0.2%
Hoop stress at interface (isotropic)	1429 MPa	1429.7 MPa	0.05%
Hoop stress at interface (orthotropic)	876 MPa	874.7 MPa	0.15%
Hoop stress at outer surface (orthotropic)	755 MPa	759.1 MPa	0.54%

TABLE 4-4: COMPARISON OF THE COMSOL MULTIPHYSICS RESULTS VS. NAFEMS TARGET FOR CASE 2.

POSITION	**COMSOL MULTIPHYSICS RESULTS CASE 2**	**NAFEMS TARGET**	**ERROR**
Hoop stress at inner surface (isotropic)	1384.3 MPa	1381 MPa	0.24%
Hoop stress at interface (isotropic)	1259 MPa	1259.6 MPa	0.05%

TABLE 4-4: COMPARISON OF THE COMSOL MULTIPHYSICS RESULTS VS. NAFEMS TARGET FOR CASE 2.

POSITION	COMSOL MULTIPHYSICS RESULTS CASE 2	NAFEMS TARGET	ERROR
Hoop stress at interface (orthotropic)	1058 MPa	1056 MPa	0.19%
Hoop stress at outer surface (orthotropic)	931 MPa	936.1 MPa	0.54%

As is evident from Table 4-3 and Table 4-4, the COMSOL Multiphysics results are in good agreement with the NAFEMS target.

References

1. *Wrapped Thick Cylinder under Pressure and Thermal Loading,* NAFEMS test No. R0031/2, Composite Benchmarks Issue 2, NAFEMS, 2001.

2. *Finite element analysis of composite materials*, I.C. Taig, NAFEMS R0003, NAFEMS, 1992.

Model Library path: `Structural_Mechanics_Module/Benchmark_Models/wrapped_cylinder`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. On the **New** page, select **3D** from the **Space Dimension** list and double-click on **Structural Mechanics Module**.
2. Select **Solid, Stress-Strain** and then **Static analysis**.
3. Click **OK** to close the Model Navigator.

GEOMETRY MODELING

You obtain the geometry by extrusion of a structured mesh: first create the geometry in 2D, then mesh it, and finally extrude the mesh in 3D.

1. In the **Draw** menu choose **Work-Plane Settings**. Select the default one by clicking **OK**.
2. In the **Options** menu choose **Axes/Grid Settings**.
3. On the **Axis** page set the axis settings **x min** as `-0.01`, **x max** as `0.03`, **y min** as `-0.01`, and **y max** as `0.03`.

4 On the **Grid** page clear the **Auto** option and set **x spacing** to `5e-3`, **Extra x** to `2.3e-2 2.7e-2`, **y spacing** to `5e-3`, and **Extra y** to `2.3e-2 2.7e-2`. When finished, click **OK**.

5 Click the **2nd Degree Bézier Curve** button in the Draw toolbar, then click at the points (0.023, 0), (0.023, 0.023), and (0, 0.023) in that order.

6 Click the **Line** toolbar button, then click at point (0, 0.025).

7 Click the **2nd Degree Bézier Curve** button, then click at the points (0.025, 0.025) and (0.025, 0) in that order.

8 Right-click to create the solid object.

Steps 5–8 create the cross section of the inner cylinder. Proceed in the same way to generate the outer cylinder with an inner radius of 25 mm and outer radius of 27 mm.

MESH GENERATION

1 In the **Mesh** menu, choose **Mapped Mesh Parameters**.

2 Click the **Boundary** tab, select Boundary 1 and Boundary 2, and select the **Constrained edge element distribution** check box. Set **Number of edge elements** to `1`.

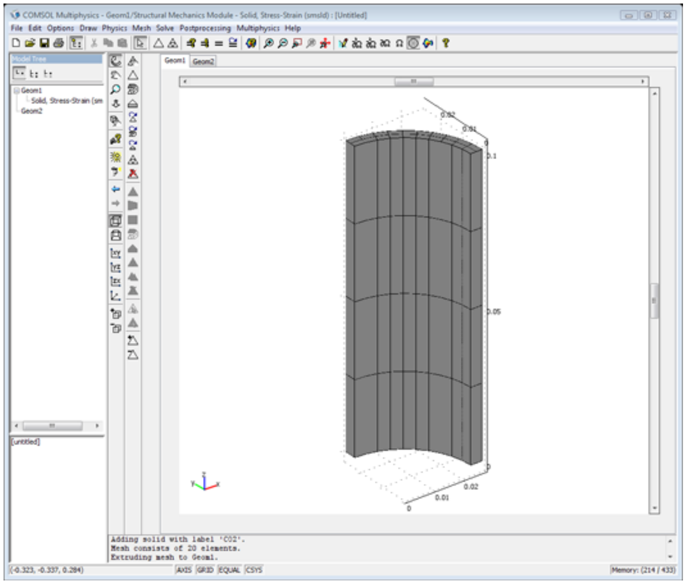
3 Select Boundary 7, select the **Constrained edge element distribution** check box, and set the **Number of edge elements** to `10`.

4 Click **Remesh**, and then click **OK**. This creates a structured 2D mesh. Next create the 3D mesh by extrusion:

5 In the **Mesh** menu, choose **Extrude Mesh**. In the **Extrusion parameters** area, type `0.1` in the **Distance** edit field.

6 Click the **Mesh** tab and type `4` in the **Number of element layers** edit field.

7 Click **OK**.



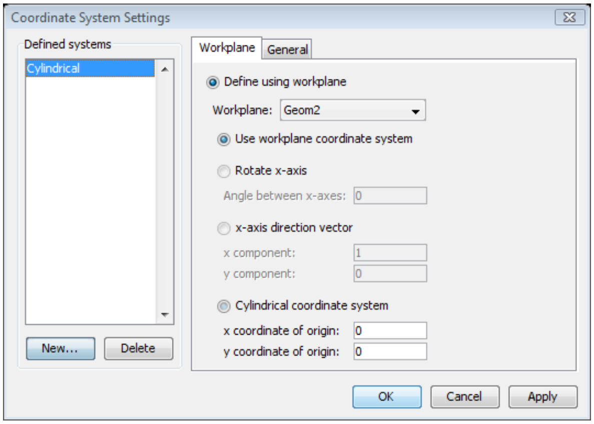
OPTIONS AND SETTINGS

In order to properly set up the orthotropic material properties regarding the material orientation, a local cylindrical coordinate system is required.

1 From the **Options** menu, choose **Coordinate Systems**.

2 Click **New** and type `Cylindrical` as the name for the new coordinate system. Click **OK**.

3 On the **Workplane** page click the **Cylindrical coordinate system** button.



4 Click **OK**.

The next step is to enter the constants that you use in the model.

1 From the **Options** menu choose **Constants** and enter constant names, expressions, and descriptions (the descriptions are optional) according to the following table:

NAME	EXPRESSION	DESCRIPTION
E1	1.3e11[Pa]	theta-component of the Young's modulus
E2	5e9[Pa]	z-component of the Young's modulus
E3	5e9[Pa]	r-component of the Young's modulus
nu12	0.25	theta-z Poisson's ratio
nu32	0	r-z Poisson's ratio
nu13	0.25	theta-r Poisson's ratio
nu31	nu13*E3/E1	r-theta Poisson's ration
G12	10e9[Pa]	theta-z shear modulus
G32	5e9[Pa]	r-z shear modulus
G31	10e9[Pa]	r-theta shear modulus

2 Click **OK**.

PHYSICS SETTINGS

Subdomain Settings

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

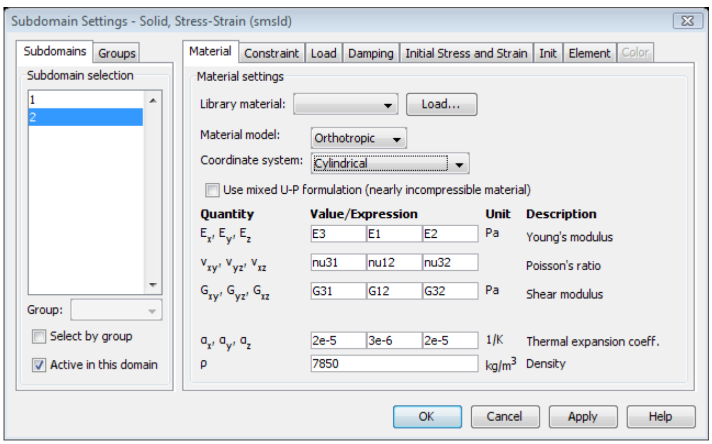
2 Select Subdomain 1 and select **Cylindrical** from the **Coordinate system** list.

3 Type `2.1e11`, `0.3`, and `2e-5` in the **Young's Modulus**, the **Poisson's ratio**, and the **Thermal expansion coeff.** edit fields, respectively.

4 Select Subdomain 2. Then select **Orthotropic material** from the **Material model** list and **Cylindrical** from the **Coordinate system** list.

5 Enter material property values according to the following table:

QUANTITY	VALUE/EXPRESSION			UNIT	DESCRIPTION
E_x, E_y, E_z	E3	E1	E2	Pa	Young's modulus
ν_{xy}, ν_{yz}, ν_{xz}	nu31	nu12	nu32		Poisson's ratio
G_{xy}, G_{yz}, G_{xz}	G31	G12	G32	Pa	Shear modulus
α_x, α_y, α_z	2e-5	3e-6	2e-5	1/K	Thermal expansion coeff.



6 Click **OK**.

Boundary Conditions

1 Open the **Boundary Settings** dialog box.

2 Select Boundary 2. On the **Load** page, select **Cylindrical** from the **Coordinate system** list, and enter `200e6` in the **Face load (force/area) xl dir.** edit field.

3 Click the **Constraint** tab. Select Boundaries 1, 3, 5, 7, 10, and 11.

4 Select **Symmetry plane** from the **Constraint condition** list.

5 Click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

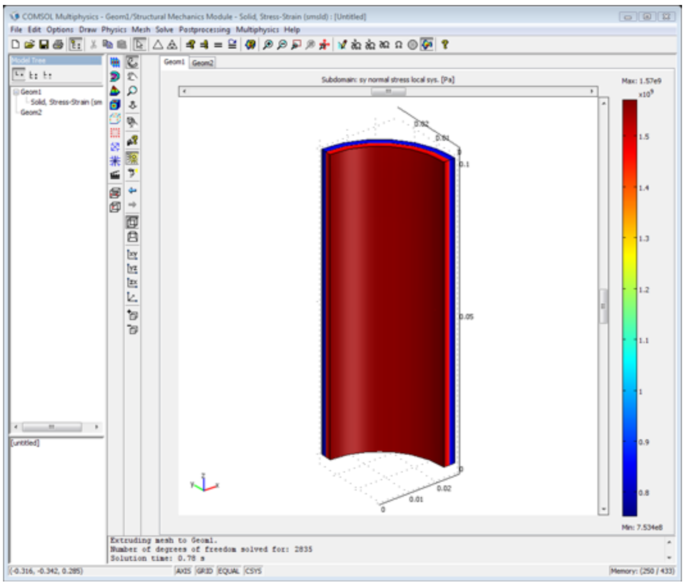
POSTPROCESSING AND VISUALIZATION

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

2 On the **General** page, clear the **Slice** check box and select the **Subdomain** check box in the **Plot type** area.

3 Click the **Subdomain** tab and select **sy normal stress local sys.** in the **Predefined quantities** list.

4 Click **OK**.



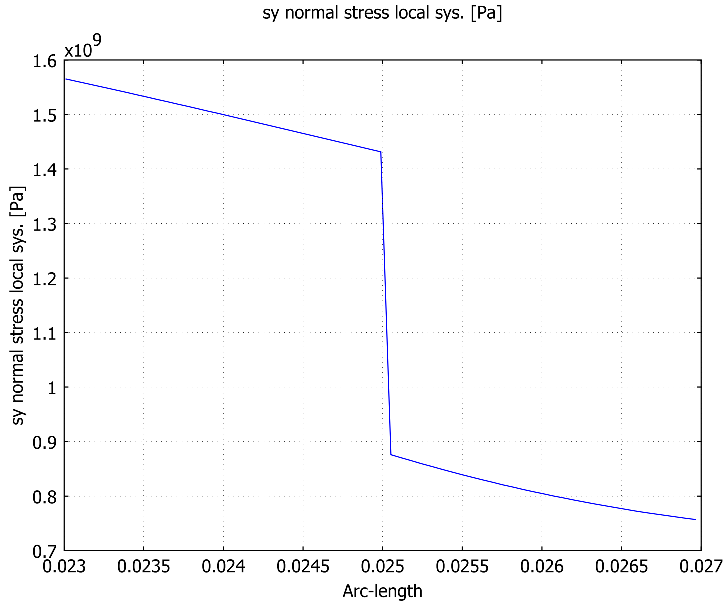
5 From the **Postprocessing** menu choose **Cross-Section Plot Parameters**. Click the **Line/ Extrusion plot** button on the **General** page.

6 Click the **Line/Extrusion** tab and select **sy normal stress local sys.** from the **Predefined quantities** list.

7 In the **Cross-section line data** area, type `27e-3` in both the **x1** and the **y1** edit field.

8 Set the **Line resolution** to `600`.

9 Click **OK**.



Do not close the cross-sectional plot; you will use it later to compare the results with the thermal loading case.

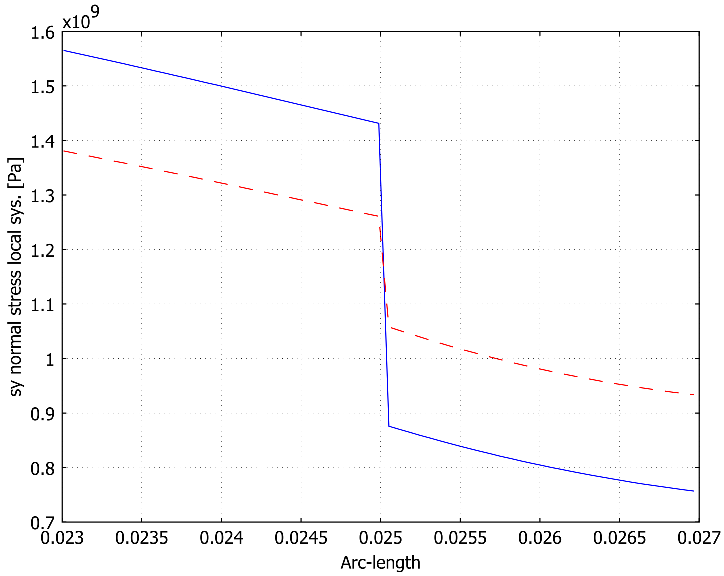
This completes the first load case of the benchmark.

SECOND CASE INCLUDING THERMAL LOADING

1. Open the **Subdomain Settings** dialog box from the **Physics** menu, and select both Subdomain 1 and Subdomain 2.
2. On the **Load** page, select the **Include thermal expansion** check box and enter `130` in the **Strain temperature** edit field.
3. Click **OK**.
4. Click the **Solve** button on the Main toolbar.
5. Open the **Cross-Section Plot Parameters** dialog box from the **Postprocessing** menu, and select the **Keep current plot** check box on the **General** page.
6. Click the **Line/Extrusion** tab; then click **Line Settings**.
7. Select **Color** and **Dashed line** for **Line color** and **Line style**, respectively.
8. Click **OK**.

9 Click **OK**.

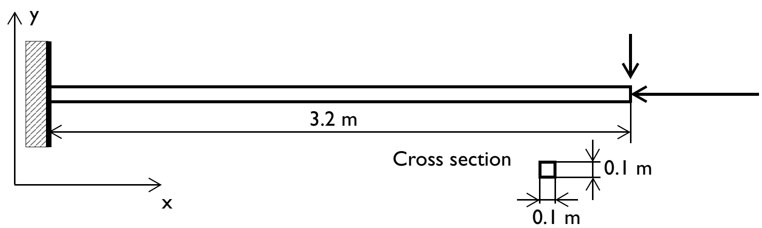
sy normal stress local sys. [Pa]



Large Deformation Beam

Model Definition

In this example you study the deflection of a cantilever beam undergoing very large deflections. The model is called "Straight Cantilever GNL Benchmark" and is described in detail in section 5.2 of NAFEMS *Background to Finite Element Analysis of Geometric Non-linearity Benchmarks* (Ref. 1).



In addition to the original problem formulation, where the complete history of the displacement is sought, you undertake a linearized buckling analysis of the structure.

This example uses the plane stress application mode.

GEOMETRY

- The length of the beam is 3.2 m.
- The cross section is a square with side lengths 0.1 m.

MATERIAL

The beam is linear elastic with $E = 2.1\cdot10^{11}$ N/m^2 and $\nu = 0$.

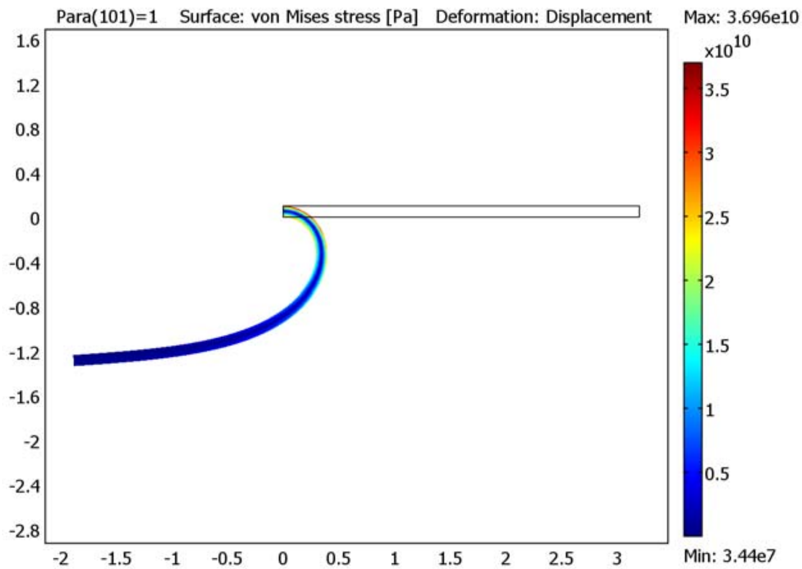
CONSTRAINTS AND LOADS

- The left end is fixed. This boundary condition is actually compatible with beam theory assumptions only in the case $\nu = 0$.
- The right end is subjected to distributed loads with the resultants $F_x = -3.844\cdot10^6$ N and $F_y = -3.844\cdot10^3$ N.

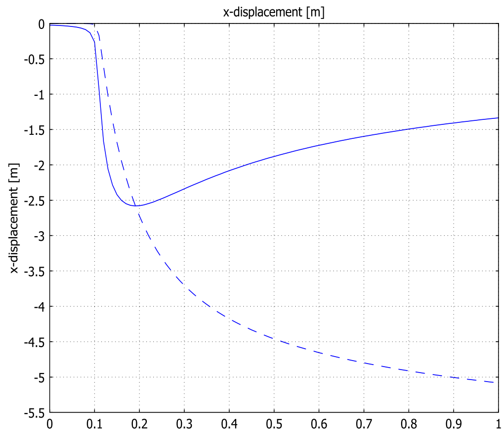
Results and Discussion

Due to the large compressive axial load and the slender geometry, this is a buckling problem. If you are to study the buckling and post buckling behavior of a symmetric problem, it is necessary to perturb the symmetry somewhat. Here the small transversal load serves this cause. An alternative approach would be to introduce an initial imperfection in the geometry.

The final state (using 1:1 displacement scaling) is shown below.



The vertical and horizontal displacements of the tip versus the scaled axial force are shown in the next graph, using solid and dashed lines, respectively.



The following table contains a summary of some significant results. Because the reference values are given as graphs, an estimate of the error caused by reading this graph is added:

RESULT	COMSOL MULTIPHYSICS	REFERENCE
Maximum vertical displacement at the tip	-2.58	-2.58 ± 0.02
Final vertical displacement at the tip	-1.33	-1.36 ± 0.02
Final horizontal displacement at the tip	-5.08	-5.04± 0.04

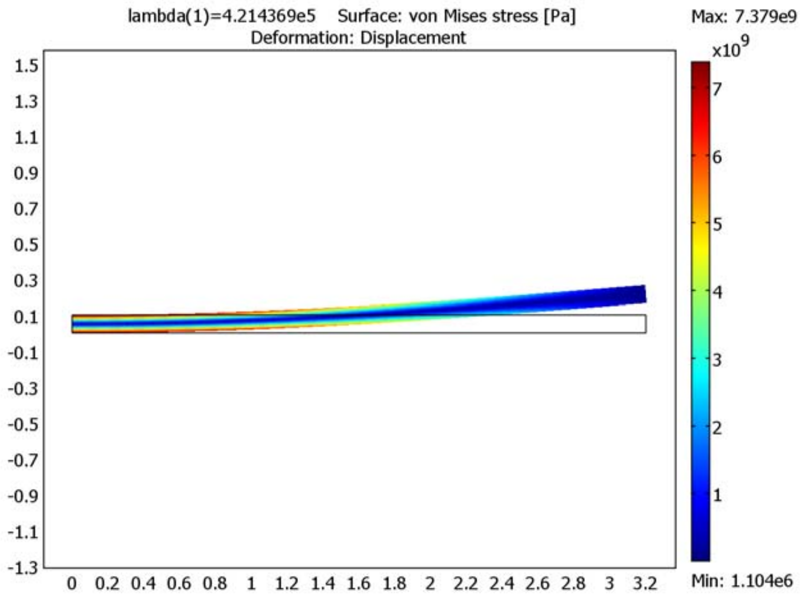
The results are in excellent agreement, especially considering the coarse mesh used.

The plot of the axial deflection reveals that an instability occurs at a parameter value close to 0.1, corresponding to the axial load $3.84 \cdot 10^5$.

This problem (without the small transverse load) is usually referred to as the Euler-1 case. The theoretical critical load is

$$P_c = \frac{\pi^2 EI}{4L^2} = \frac{\pi^2 \cdot 2.1 \cdot 10^{11} \cdot \frac{0.1^4}{12}}{4 \cdot 3.2^2} = 4.22 \cdot 10^5 \text{ N} \qquad (4\text{-}1)$$

The critical buckling load computed using COMSOL Multiphysics is $4.21 \cdot 10^5$ N, very close to the theoretical value. The corresponding buckling mode shape is shown below.



It is often seen in practice that the critical load of an imperfect structure is significantly lower than that of the ideal structure.

Reference

1. Becker, A. A., *Background to Finite Element Analysis of Geometric Non-linearity Benchmarks*, NAFEMS, Ref: -R0065, Glasgow.

Model Library path: `Structural_Mechanics_Module/Benchmark_Models/large_deformation_beam`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **2D** in the **Space dimension** list on the **New** page in the **Model Navigator**.

2 Select **Structural Mechanics Module**>**Plane Stress**>**Parametric analysis**.

3 Click **OK**.

OPTIONS AND SETTINGS

Select **Axes/Grid Settings** from the **Options** menu and specify axis and grid settings according to the following table:

AXIS		GRID	
x min	`-0.1`	x spacing	`0.1`
x max	`3.3`	Extra x	
y min	`-0.1`	y spacing	`0.1`
y max	`0.2`	Extra y	

GEOMETRY MODELING

Click the **Rectangle/Square** toolbar button and click the left mouse button at (0, 0). Move the mouse to (3.2, 0.1) and click the left mouse button again.

PHYSICS SETTINGS

Boundary Settings

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

2 Specify boundary settings according to the following tables:

BOUNDARY	1	
Page	Constraint	
	Constraint condition	Fixed

BOUNDARY	4	
Page	Load	
	F_x	`-3.844e6/0.1*Para`
	F_y	`-3.844e3/0.1`

Both forces are specified as forces/length (this is the reason for dividing with 0.1). The horizontal force is multiplied with the parameter `Para`, which increases the force using the parametric solver.

Subdomain Settings

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

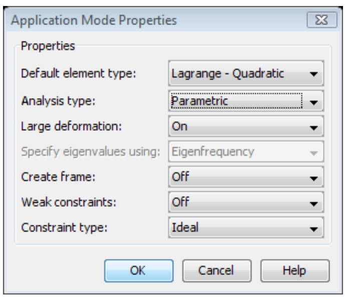
2 Specify material data according to the following table

SUBDOMAIN	1	
Page	Material	
	E	2.1e11
	ν	0
	thickness	0.1

Large Deformation Property

1 Select **Properties** from the **Physics** menu.

2 From the **Large deformation** list select **On**.



MESH GENERATION

Click the **Initialize Mesh** toolbar button to generate the mesh.

COMPUTING THE SOLUTION

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

2 Enter the name Para in the **Parameter name** edit field.

3 Type `0:0.01:1` in the **Parameter values** edit field.

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

5 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

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

2 Click the **Deform** tab.

3 Select the **Deformed shape plot** check box.

4 Clear the **Auto** check box, then set the **Scale factor** to 1. Click **OK**.

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

6 Select **Cross-Section Plot Parameters** from the **Postprocessing** menu.

7 Select all parameter values in the **Solutions to use** list.

8 Check **Keep current plot**.

9 Click the **Point** tab.

10 Enter the coordinates x = 3.2 and y = 0.05 to plot the solution at the beam end.

11 Select **y-displacement** in the **Predefined quantities** list, then click **Apply**.

12 Click the **Line Settings** button, then select **Dashed line** in the **Line style** list. Click **OK**.

13 Select **x-displacement** in the **Predefined quantities** list, then click **OK**.

COMPUTING THE LINEARIZED BUCKLING LOAD

A linear buckling analysis consists of two steps: First you apply the load (often a unit load) and run a static analysis. Then you start a second analysis, solving an eigenvalue problem. The result is a critical load factor causing instability of the structure.

Static Preload

1 Select **Properties** from the **Physics** menu.

2 Select **Static** from the **Analysis type** list.

3 Select **Off** from the **Large deformation** list, then click **OK**.

4 Select **Boundary Settings** from the **Physics** menu and change the loads according to the following table. When finished, click **OK**.

BOUNDARY	4			
Page	Load			
	F_x	-1/0.1	F_y	0

5 Click the **Solve** button on the Main toolbar.

Buckling Analysis

1 Select **Properties** from the **Physics** menu.

2 Select **Linear buckling** from the **Analysis type** list, then click **OK**.

3 Select **Solver Parameters** from the **Solve** menu.

4 Enter 1 in the **Desired number of critical load factors** edit field.

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

6 Click the **Restart** button on the Main toolbar.

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

8 Click the **Deform** tab.

9 Select the **Auto** check box under **Scale factor**, then click **OK**.

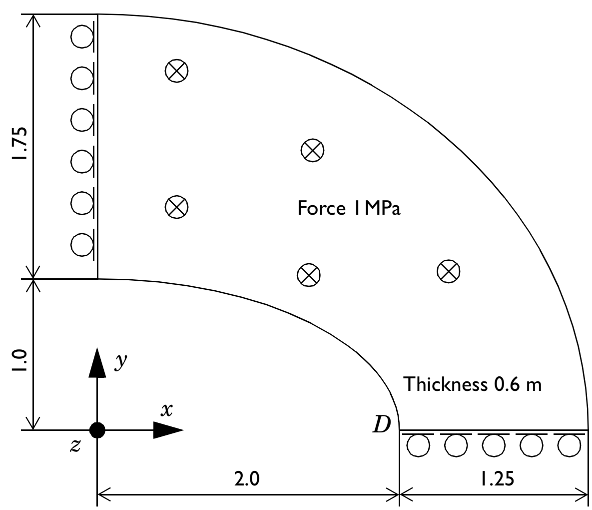
Thick Plate Stress Analysis

In the following example you build and solve a solid mechanics model using the Solid, Stress-Strain application mode.

This model describes the static stress analysis of a simple geometry. The model is NAFEMS Test No LE10, "Thick Plate Pressure" described on page 77 in NAFEMS Background to Benchmarks (Ref. 1). The computed stress level is compared with the values given in the benchmark report.

Model Definition

The geometry is an ellipse with an ellipse shaped hole in it. Due to symmetry in load and geometry, the analysis only includes a quarter of the ellipse.



MATERIAL

Isotropic with, E=2.1·10^{11} Pa, ν=0.3.

LOAD

A distributed force of 10^6 MPa on the upper surface pointing in the negative z direction.

CONSTRAINTS

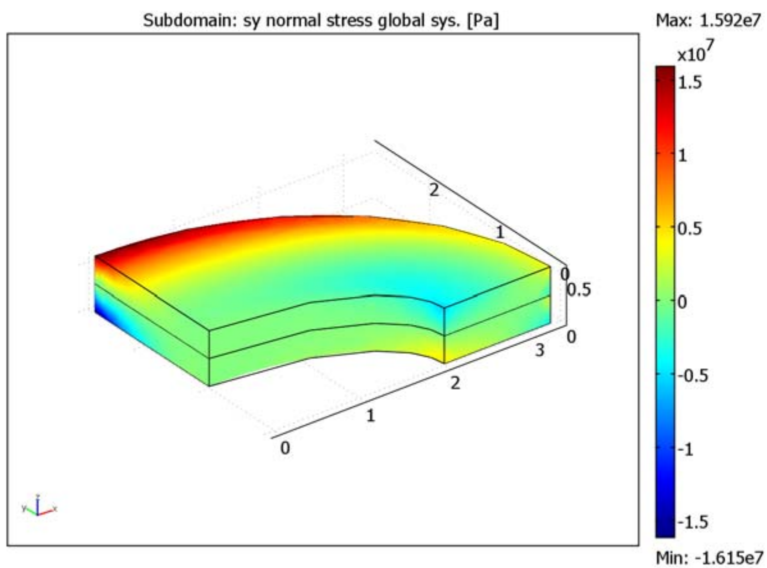
- Symmetry planes, $x = 0$, $y = 0$.
- Outer surface constrained in the x and y direction.
- Mid plane on outer surface constrained in the z direction.

Results

The normal stress σ_y on the top surface at the inside of the elliptic hole (marked with D (2, 0, 0.6) in the figure) is in close agreement with the NAFEMS benchmark (Ref. 1). The coordinates of D are (2, 0, 0.6).

RESULT	COMSOL MULTIPHYSICS	NAFEMS (REF. 1)
σ_y (at D)	-5.45 MPa	-5.38 MPa

A COMSOL Multiphysics plot visualizing the stress level is shown below.



Reference

1. Davies, G. A. O., Fenner, R. T., and Lewis, R. W., *Background to Benchmarks*, NAFEMS, Glasgow, 1993.

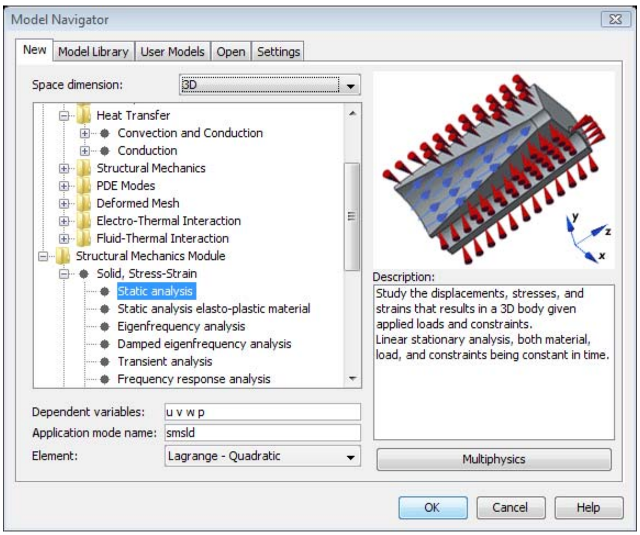
Model Library path: `Structural_Mechanics_Module/Benchmark_Models/thick_plate`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **3D** in the **Space dimension** list on the **New** page in the **Model Navigator**.

2 Select **Structural Mechanics Module>Solid, Stress-Strain>Static analysis** and click **OK.**



GEOMETRY MODELING

The quarter of the ellipse is drawn in a work plane and extruded into a 3D solid.

1 Select **Work-Plane Settings** from the **Draw** menu.

2 Click **OK** in the **Work-Plane Settings** dialog box to create a work plane using the default settings, global xy-plane at $z = 0$.

3 Select **Axes/Grid Settings** from the **Options** menu, clear **Auto** on the **Grid** page, and give axis and grid settings according to the following table:

AXIS		GRID	
x min	-4	x spacing	0.25
x max	4	Extra x	
y min	-3	y spacing	0.25
y max	3	Extra y	

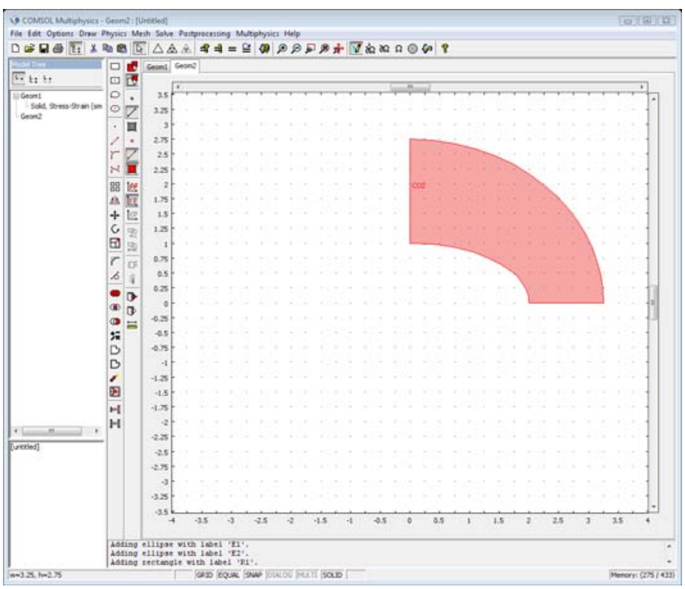
4 Click the **Ellipse/Circle (Centered)** button on the Draw toolbar and click the left mouse button at (0, 0) move the mouse to (2, 1) and click the left mouse button again to create the inner ellipse.

5 Click the **Ellipse/Circle (Centered)** button on the Draw toolbar and click the left mouse button at (0, 0) move the mouse to (3.25, 2.75) and click the left mouse button again to create the outer ellipse.

6 Select both ellipses and click the **Difference** button on the Draw toolbar to cut a hole.

7 Draw a rectangle with bottom left corner at (0, 0) and top right corner at (3.25, 2.75).

8 Select the ellipse with a hole in and the rectangle, and then click the **Intersection** button on the Draw toolbar to create a quarter of the ellipse.



Now you need to extrude the quarter of the ellipse to create a 3D solid object.

9 Select the quarter and then choose **Extrude** from the **Draw** menu.

10 Enter `0.3` as **Distance** in the **Extrude** dialog box and click **OK**.

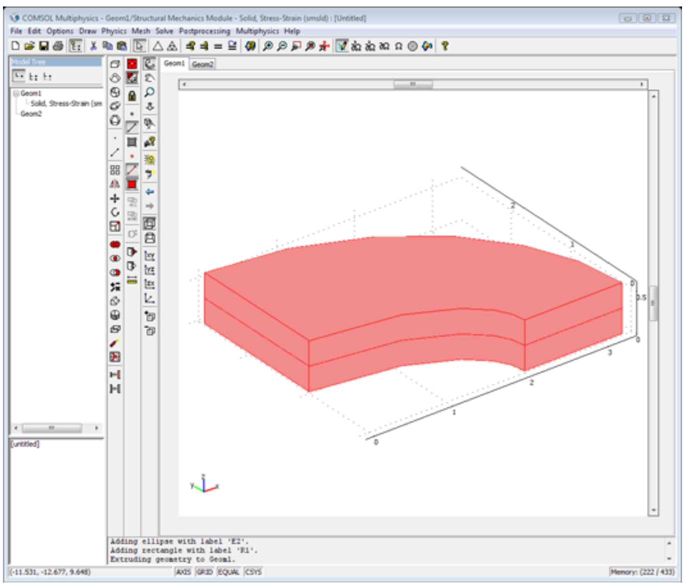
11 Click the **Array** toolbar button to make a copy of the object.

12 Enter `0.3` as **z Displacement** and `2` as **z Array size** in the **Array** dialog box and click **OK**.

13 In order to remove interior boundaries but keep edges needed for the boundary condition, click the **Create Composite Object** button on the Draw toolbar.

14 Select **EXT1** and **EXT2** in the **Object selection** list, clear the **Keep interior borders** check box and click the **Keep interior edges** check box in the **Create Composite Object** dialog box and click **OK**.

15 Click the **Zoom Extents** button on the Main toolbar to see the finished 3D object.



PHYSICS SETTINGS

In this section the analysis type is specified, and the edge, boundary, and subdomain settings is made.

Application Mode Properties

The analysis type is controlled from the **Application Mode Properties** dialog box, opened from **Properties** on the **Physics** menu. Static analysis was selected already in the **Model Navigator** so there is no need to change the analysis type.

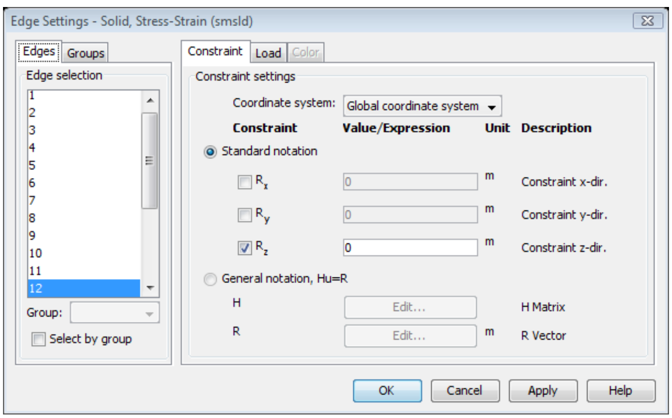
Edge Settings

Constrain the z-displacement along the midplane edge of the outer surface.

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

2 Specify edge settings according to the following table:

SETTINGS	EDGE 12	
Page	Constraint	
	R_z	0



Boundary Settings

Constrain the symmetry planes normal to the surface and the outer surface in the x and y directions. Specify the distributed force on the top surface to -10^6 in the z direction.

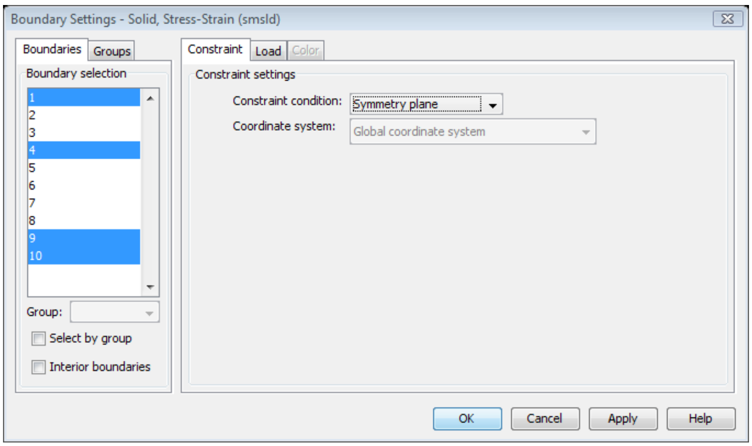
1 Select **Boundary Settings** from the **Physics** menu.

2 Specify boundary settings according to the following tables:

SETTINGS	BOUNDARIES 1, 4, 9, 10		BOUNDARIES 7, 8	
Page	Constraint		Constraint	
	Constraint condition	Symmetry plane	Constraint condition	Prescribed displacement
			Coordinate system	Global coordinate system

SETTINGS	BOUNDARIES 1, 4, 9, 10		BOUNDARIES 7, 8	
			R_x	0
			R_y	0

SETTINGS	BOUNDARY 6	
Page	Load	
	F_z	-1E6



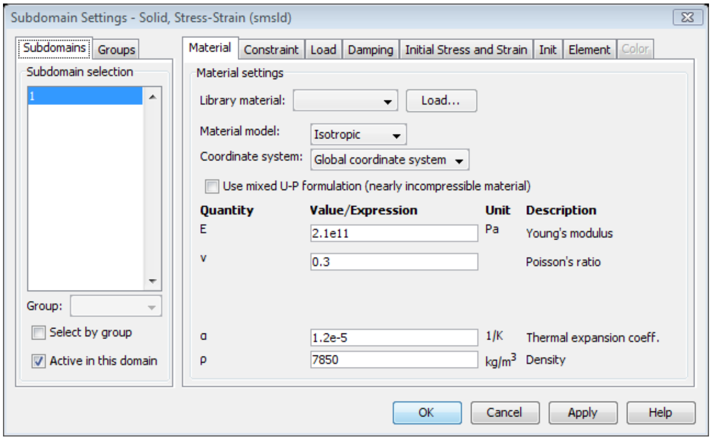
Subdomain Settings

Specify the material properties of the plate.

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

2 Specify subdomain settings according to the following table:

SETTINGS	SUBDOMAIN 1	
Page	Material	
	Material model	Isotropic material
	E	2.1E11
	ν	0.3



MESH GENERATION

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

2 Select `Fine` from the **Predefined mesh sizes** list, click **OK** to close the dialog box.

3 Click the **Initialize Mesh** toolbar button to mesh the geometry.

COMPUTING THE SOLUTION

Click the **Solve** toolbar button in order to compute the solution.

POSTPROCESSING AND VISUALIZATION

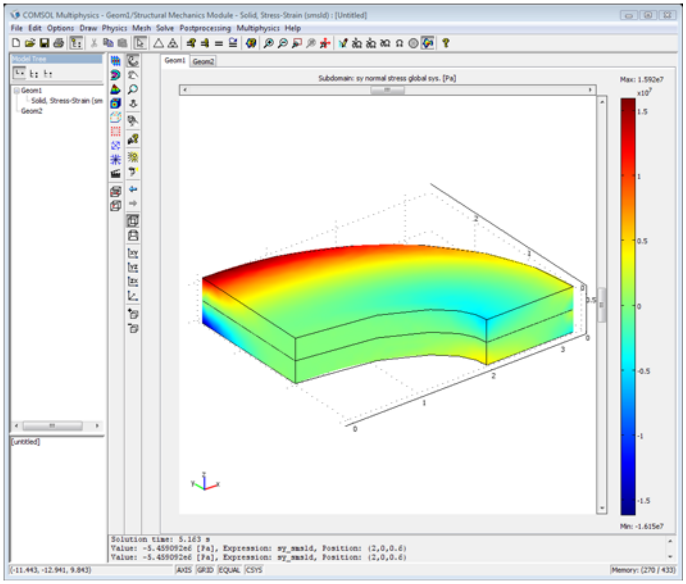
Analyze the global σ_y stress at D.

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

2 On the **General** page, clear the **Slice** check box and select the **Subdomain** check box in the **Plot type** area

3 Select **sy normal stress global sys.** from the **Predefined quantities** list on the **Subdomain** page.

4 Click **OK** to close the dialog box and view the plot.

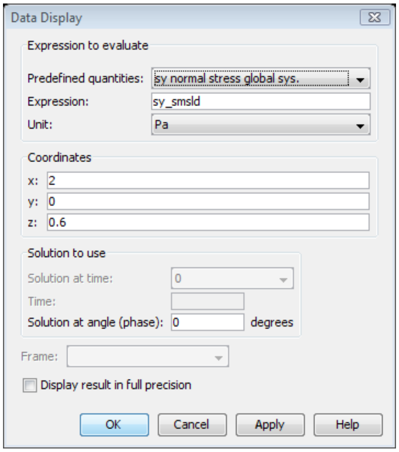


Looking at the plot above the stress level at D seem to be around $-5 \cdot 10^6$. To get a more accurate value use the **Data Display** dialog box.

5 Select **Data Display>Subdomain** from the **Postprocessing** menu to open the **Data Display** dialog box.

6 Enter the coordinates of D in the **x**, **y**, and **z** coordinates edit fields. The coordinates are (2, 0, 0.6).

7 Select **sy normal stress global sys.** from the **Predefined quantities** list.



8 Click **OK** to display the stress. The value appears in the message log. The stress value is -5.45 MPa, which is in close agreement with the NAFEMS value.

Kirsch Infinite Plate Problem

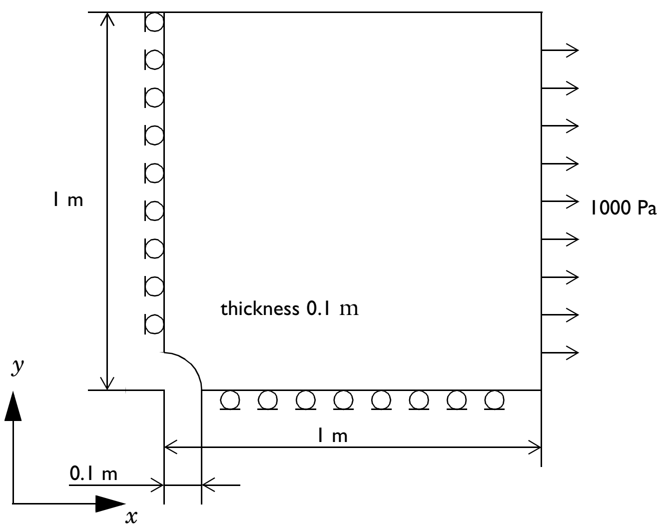
In the following example you will build and solve a model using the Plane Stress application mode.

This model describes the static stress analysis of a simple geometry, a small hole in an infinite plate. The model is a classic benchmark, and the theoretical solution was derived by G. Kirsch in 1898 (see for example D. Roylance, Mechanics of Material, Wiley, 1996 4.) The stress level is compared with the theoretical values.

Model Definition

The model is the Kirsch plate described on page 184 in D. Roylance *Mechanics of Materials* (Ref. 1).

The infinite plate is modeled as a 2 m-by-2 m plate with a hole with a radius of 0.1 m in the middle. Due to symmetry in load and geometry only a quarter of the plate is modeled.



MATERIAL

Isotropic material with, $E = 2.1 \cdot 10^{11}$ Pa, $\nu = 0.3$.

LOAD

A distributed force of 10^3 Pa on the right edge pointing in the x direction.

CONSTRAINTS

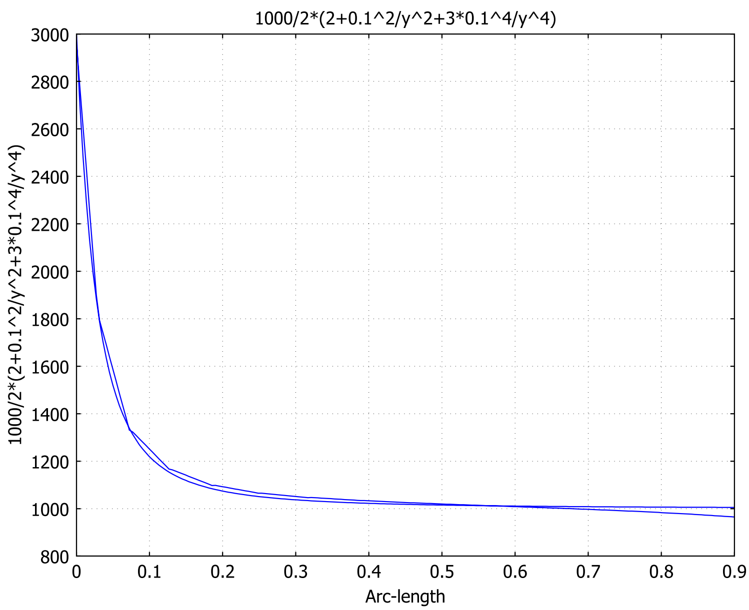
Symmetry planes, $x = 0$, $y = 0$.

Results

The normal stress σ_x is plotted as a function of the y-coordinate along the left symmetry edge. The theoretical stress according to D. Roylance (Ref. 1) is

$$\sigma_x = \frac{1000}{2}\left(2 + \frac{0.1^2}{y^2} + 3\frac{0.1^4}{y^4}\right)$$

In the following plot the theoretical values are plotted as a comparison.



The theoretical values from Ref. 1 are in close agreement with the result from COMSOL Multiphysics.

Reference

1. Roylance, D., *Mechanics of Materials*, Wiley, 1996.

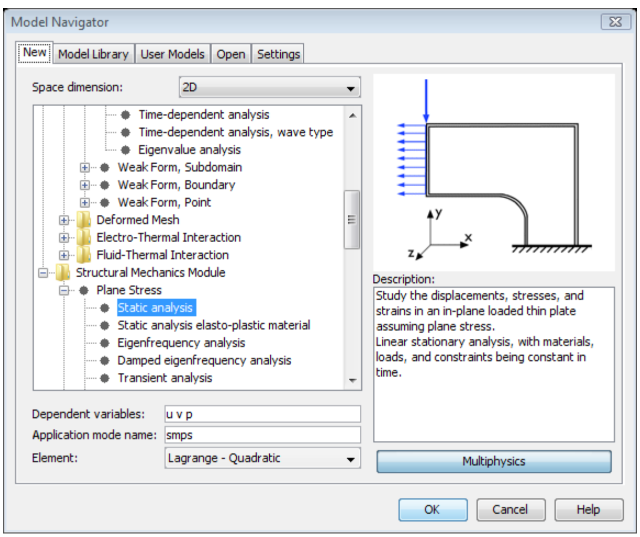
Model Library path: `Structural_Mechanics_Module/Benchmark_Models/kirsch_plate`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **2D** in the **Space dimension** list on the **New** page in the **Model Navigator**.

2 Select **Structural Mechanics Module>Plane Stress>Static analysis** and click **OK**.



OPTIONS AND SETTINGS

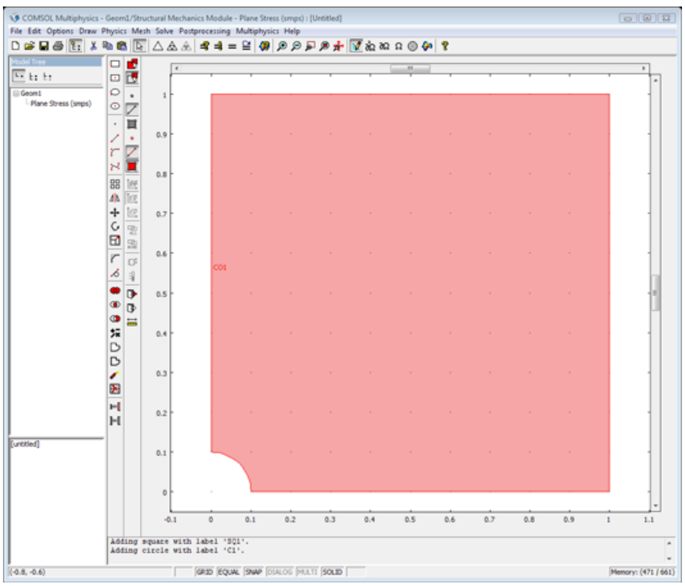
Select **Axes/Grid Settings** from the **Options** menu and give axis and grid settings according to the following table:

AXIS		GRID	
x min	`-0.5`	x spacing	`0.1`
x max	`1.5`	Extra x	

AXIS		GRID	
y min	-0.5	y spacing	0.1
y max	1.5	Extra y	

GEOMETRY MODELING

1. Click the **Rectangle/Square** toolbar button and click the left mouse button at (0, 0). Press Ctrl while moving the mouse to (1, 1) and click the left mouse button again to create a square.
2. Click the **Ellipse/Circle (Centered)** toolbar button and click the left mouse button at (0, 0) press Ctrl while moving the mouse to (0.1, 0.1) and click the left mouse button again to create a centered circle.
3. Select the square and circle and click the **Difference** toolbar button to cut a hole in the plate.



PHYSIC SETTINGS

Boundary Settings

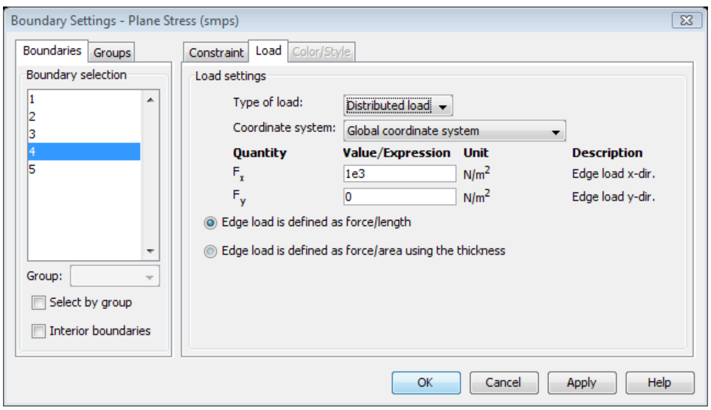
Constrain the symmetry edges in the x and y directions. Specify the distributed force on the right edge to 10^3 in the x direction.

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

2 Specify boundary settings according to the following tables:

	BOUNDARIES 1, 3	
Page	Constraint	
	Constraint condition	Symmetry plane

	BOUNDARY 4	
Page	Load	
	F_x	`1e3`
	Load def.	`force/area`



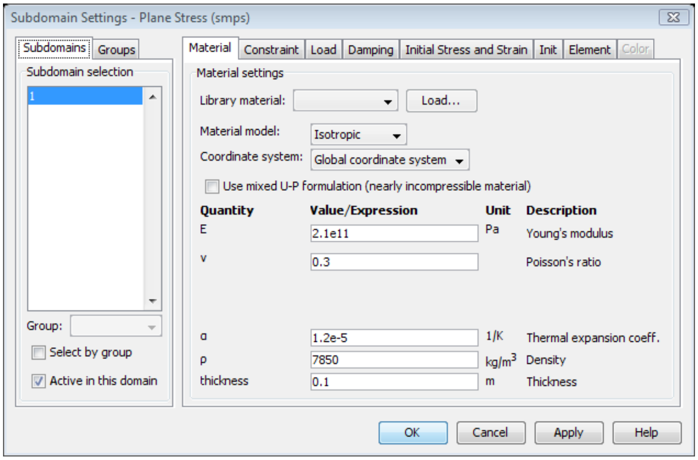
Subdomain Settings

Specify the material properties of the plate.

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

2 Specify subdomain settings according to the following table:

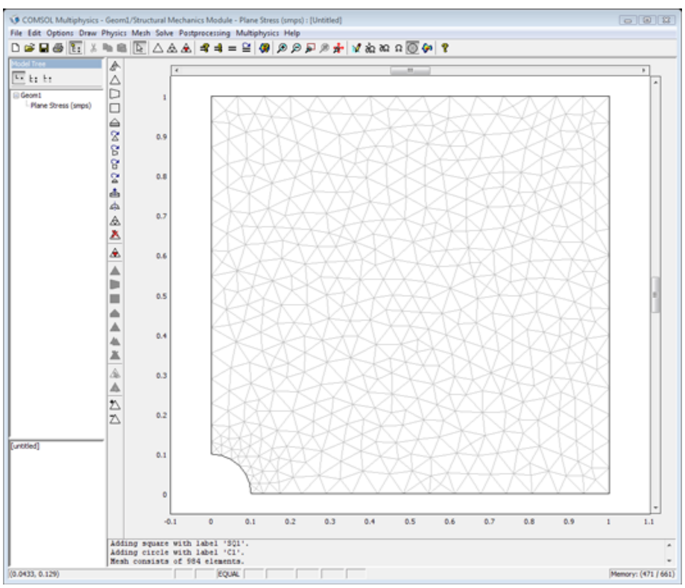
	SUBDOMAIN 1	
Page	Material	
	Material model	Isotropic material
	E	`2.1E11`
	ν	`0.3`
	thickness	`0.1`



MESH GENERATION

Use the default mesh settings.

1 Click the **Initialize Mesh** toolbar button on the main toolbar to create the mesh.



COMPUTING THE SOLUTION

1 Click the **Solve** toolbar button in order to compute the solution.

POSTPROCESSING AND VISUALIZATION

Analyze the global σ_x stress along the symmetry line x=0 between y coordinates 0.1 and 1.0 and compare it with the analytical solution for the infinite plate.

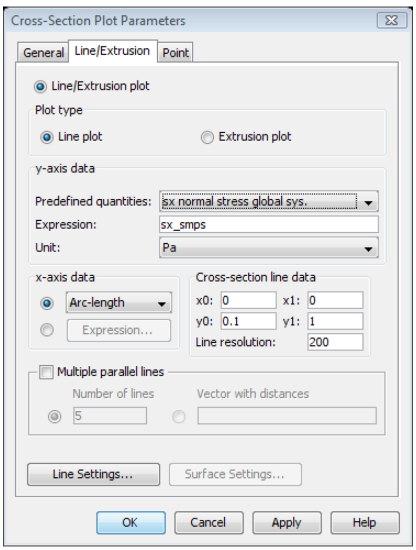
1 Select **Cross-Section Plot Parameters** from the **Postprocessing** menu.

2 Click the **Line/Extrusion** tab.

3 Select **Plane Stress>sx normal stress global sys.** from the **Predefine quantities** list in **y-axis data**.

4 Type `0` and `0.1` in the **X0** and **Y0** edit fields.

5 Type `0` and `1.0` in the **X1** and **Y1** edit fields.



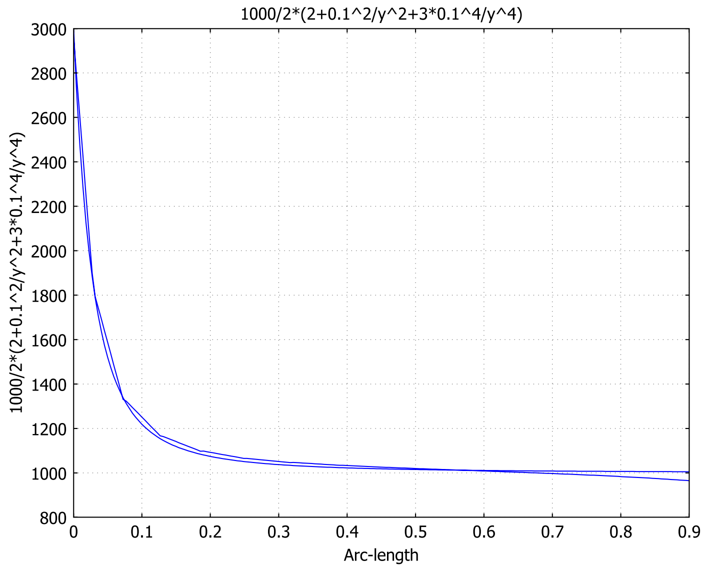
6 Click **Apply** to view the stress in an xy-diagram.

7 Click the **General** tab and select the **Keep current plot** check box to plot the analytical solution in the same diagram.

8 Click the **Line/Extrusion** tab.

9 Type `1000/2*(2+0.1^2/y^2+3*0.1^4/y^4)` in the **Expression** field.

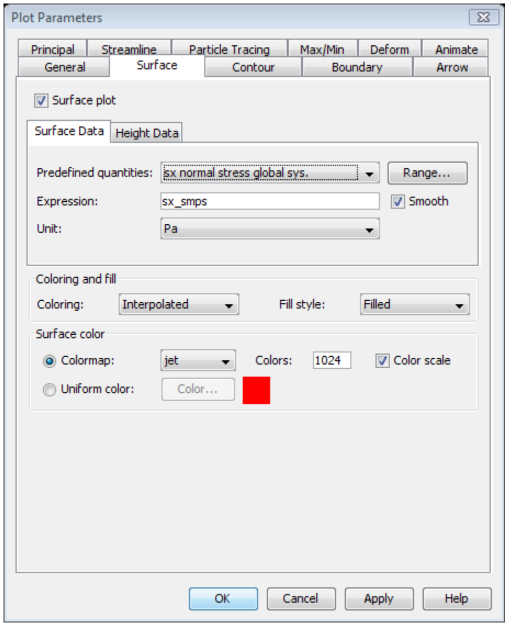
10 Click **OK** to add the analytic stress to the *x*-*y* diagram.

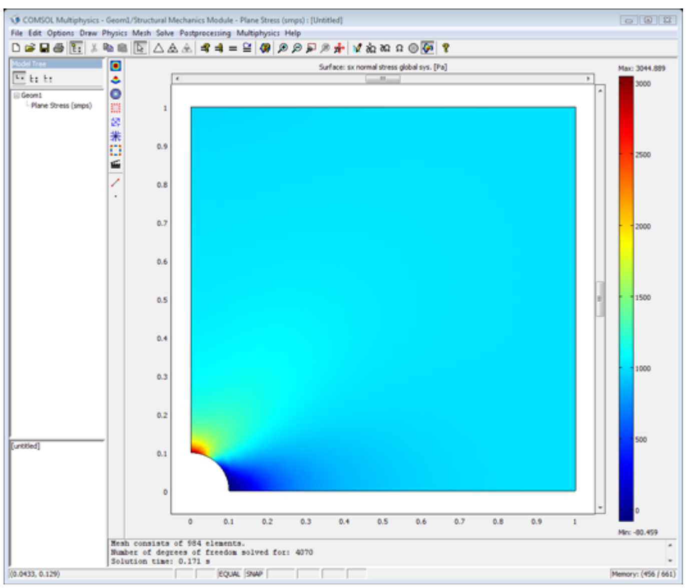


View the σ_x normal stress in a surface plot.

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

12 Select **sx normal stress global sys.** from the **Predefined quantities** list and click **OK.**





Thick Wall Cylinder Benchmark Problem

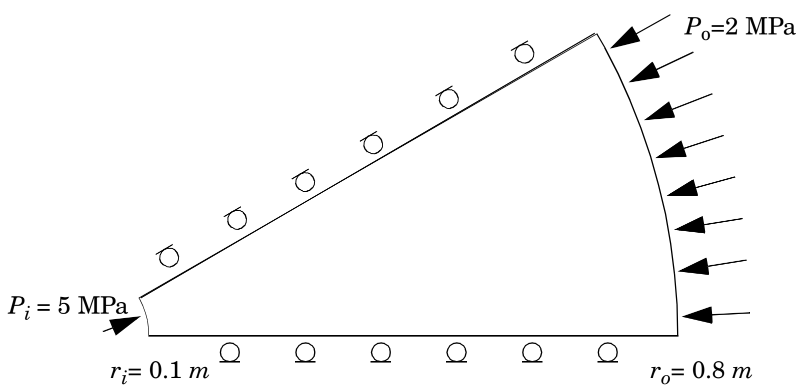
In the following example you build and solve a model using the Plane Strain application mode.

The model is a thick-walled cylinder exposed to both internal and external pressure as well as a thermal load.

The problem is both plane and axisymmetric and has an analytical solution, which can be found in Abdel-Rahman Ragab (Ref. 1).

Model Definition

Both ends of the cylinder are constrained from moving in the axial direction resulting in plane strain conditions. Due to symmetry both in load and geometry it is sufficient to model a slice of the pipe.



MATERIAL

Isotropic material with $E = 2.1 \cdot 10^{11}$ Pa, $\nu = 0.3$, and $\alpha = 1.2 \cdot 10^{-5}$ °C^{-1}.

PRESSURE LOADS

An internal pressure of $5 \cdot 10^6$ Pa and an external pressure of $2 \cdot 10^6$ Pa.

THERMAL LOAD

- Thermal strain temperature distribution from analytic solution of thermal problem. Inside temperature T_i = 500 °C, outside temperature T_o = 20 °C. The analytic solution of thermal problem from Ref. 1 is

$$T(r) = \frac{T_i - T_o}{\ln\left(\frac{r_0}{r_i}\right)}\ln\left(\frac{r_0}{r}\right) + T_o$$

- Thermal strain reference temperature: 20 °C.

CONSTRAINTS

Symmetry plane constraint condition on the symmetry planes.

Results

The stresses at the inside and outside of cylinder are compared with the analytic solutions according to Abdel-Rahman Ragab (Ref. 1) in the following table.

STRESS COMPONENT	COMSOL INSIDE	TARGET INSIDE	COMSOL OUTSIDE	TARGET OUTSIDE
σ_x (σ_r)	$-5.0 \cdot 10^7$	$-5.0 \cdot 10^7$	$-2.0 \cdot 10^7$	$-2.0 \cdot 10^7$
σ_y (σ_θ)	$-1.32 \cdot 10^9$	$-1.33 \cdot 10^9$	$3.69 \cdot 10^8$	$3.69 \cdot 10^8$
σ_z	$-1.63 \cdot 10^9$	$-1.62 \cdot 10^9$	$1.05 \cdot 10^8$	$1.05 \cdot 10^8$

The stresses are in close agreement having a maximum relative error of 0.1%.

Reference

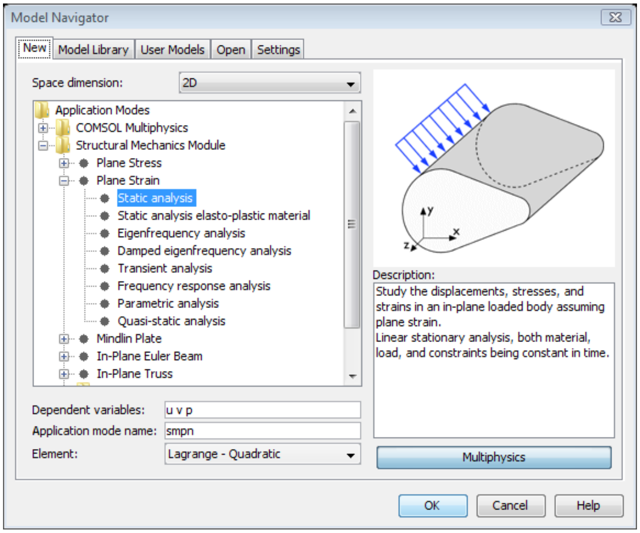
1. Abdel-Rahman Ragab, Salah Eldin Bayoumi, *Engineering Solid Mechanics*, CRC Press, 1998.

Model Library path: `Structural_Mechanics_Module/Benchmark_Models/thick_wall_cylinder`

MODEL NAVIGATOR

1 Select **2D** in the **Space dimension** list on the **New** page in the **Model Navigator.**

2 Select **Structural Mechanics Module>Plane Strain>Static analysis** and click **OK.**



OPTIONS AND SETTINGS

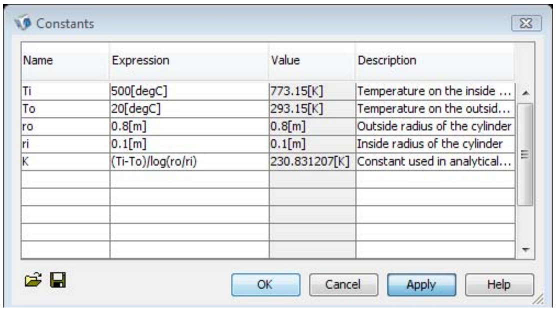
1 Select **Axes/Grid Settings** from the **Options** menu, clear **Auto** on the **Grid** page, and give axis and grid settings according to the following table:

AXIS		GRID	
x min	-0.1	x spacing	0.1
x max	1.0	Extra x	
y min	-0.1	y spacing	0.1
y max	1.0	Extra y	

2 Select **Constants** from the **Options** menu and enter constant names, expressions, and descriptions (optional) according to the following table; when done, click **OK**.

NAME	EXPRESSION	COMMENTS
Ti	500[degC]	Temperature on the inside of the cylinder
To	20[degC]	Temperature on the outside of the cylinder

NAME	EXPRESSION	COMMENTS
`ri`	`0.1[m]`	`Inside radius of the cylinder`
`ro`	`0.8[m]`	`Outside radius of the cylinder`
`K`	`(Ti-To)/log(ro/ri)`	`Constant used in analytical strain temperature expression`



GEOMETRY MODELING

1 Choose **Specify Objects>Circle** from the **Draw** menu.

2 Enter `0.8` as the **Radius** in the **Circle** dialog box and click **OK** to create the circle for the outside of the cylinder.

3 Choose **Specify Objects>Circle** from the **Draw** menu.

4 Enter `0.1` as the **Radius** in the **Circle** dialog box and click **OK** to create the circle for the inside of the cylinder.

5 Select both circles and click the **Difference** toolbar button to cut a hole in the large circle.

6 Click the **Line** toolbar button and click the left mouse button at $(0, 0)$. Move the mouse to $(1, 0)$ and click the left mouse button followed by the right mouse button to create a line.

7 Select the line and select **Copy** from the **Edit** menu.

8 Select **Paste** from the **Edit** menu.

9 Click the **Rotate** toolbar button to open the **Rotate** dialog box.

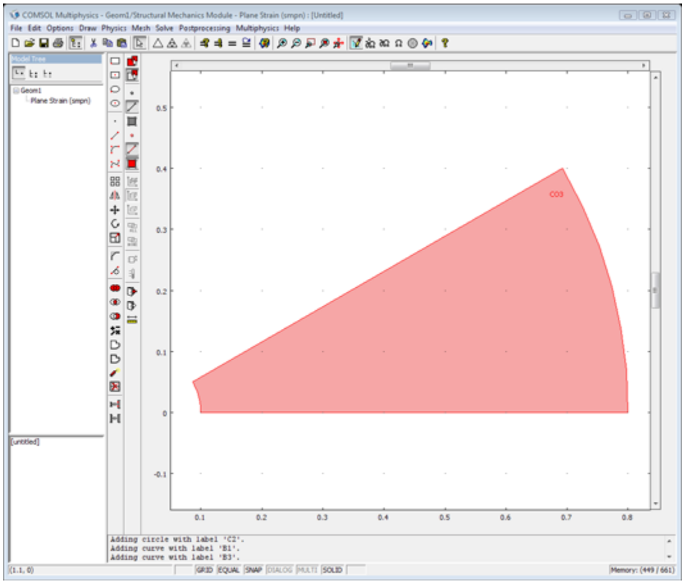
10 Enter 30 as the rotation angle and click **OK**.

11 Click the **Line** toolbar button and click the left mouse button at the right end of the first line. Move the mouse to the right end of the rotated line and click the left mouse button followed by the right mouse button to create a third line.

12 Select all three lines drawn and click the **Coerce to Solid** button on the Draw toolbar.

13 Select all objects and click the **Intersection** button on the Draw toolbar.

14 Click the **Zoom Extents** button on the Main toolbar to look at the finished geometry.



PHYSICS SETTINGS

Boundary Settings

Constrain the symmetry edges in the normal direction. Specify the internal and external pressure.

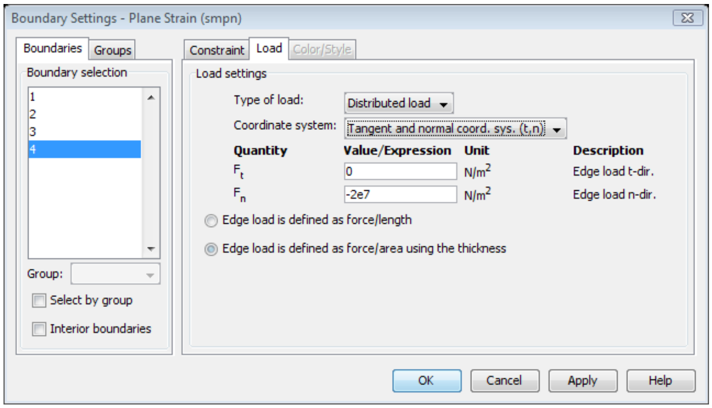
1 Select **Boundary Settings** from the **Physics** menu.

2 Specify boundary settings according to the following tables:

	BOUNDARIES 1, 2	
Page	Constraint	
	Constraint condition	Symmetry plane

	BOUNDARY 3		**BOUNDARY 4**	
Page	Load		Load	
	Coordinate system	Tangent and normal coord. sys. (t, n)	Coordinate system	Tangent and normal coord. sys. (t, n)

	BOUNDARY 3		BOUNDARY 4	
	F_n	-5e7	F_n	-2e7
	Load def.	force/area	Load def.	force/area



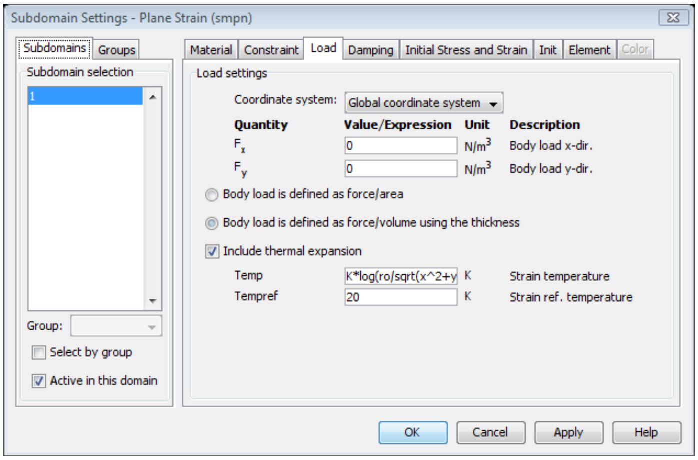
Subdomain Settings

Specify the material properties and thermal loading of the cylinder slice.

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

2 Specify subdomain settings according to the following table:

	SUBDOMAIN 1		SUBDOMAIN 1	
Page	Material		Load	
	Material model	Isotropic material	Include thermal expansion	√
	E	2.1E11	Temp	`K*log(ro/sqrt(x^2+y^2))+20`
	ν	0.3	Tempref	20

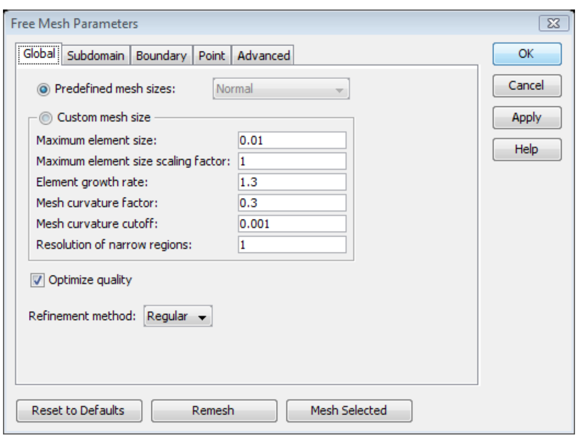


MESH GENERATION

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

2 Click the **Custom mesh size** button.

3 Type `0.01` in the **Maximum element size** edit field and click **OK**.



4 Click the **Initialize Mesh** toolbar button to create and plot the mesh.

COMPUTING THE SOLUTION

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

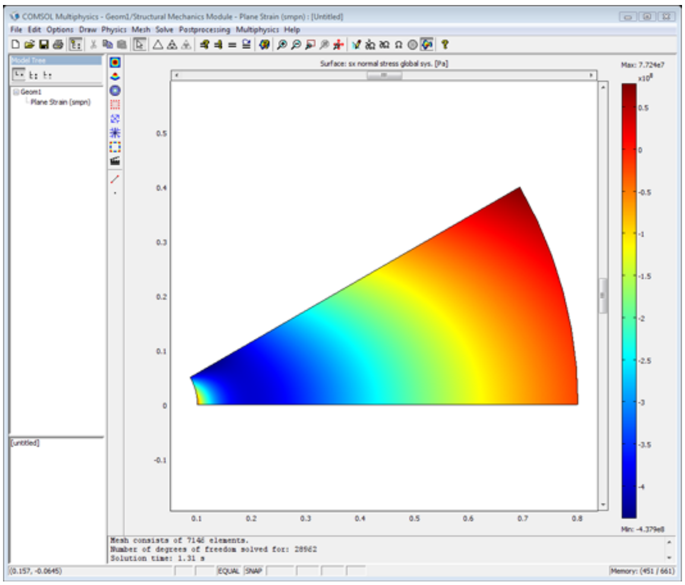
POSTPROCESSING AND VISUALIZATION

Look at the stress levels on the inside and outside of the cylinder and compare with the analytical values.

View the σ_x normal stress in a surface plot.

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

2 Click the **Surface** tab, and select **Plane Strain>sx normal stress global sys.** from the **Predefined quantities** list under **Surface Data** and click **OK**.

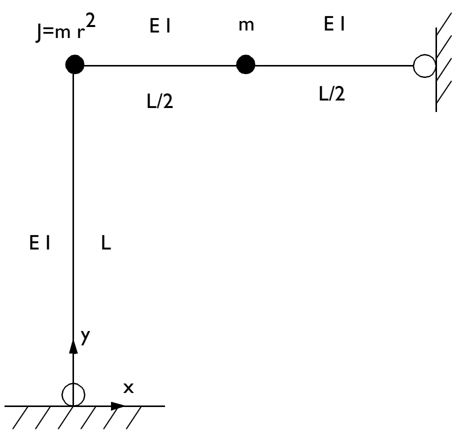


In-Plane Framework with Discrete Mass and Mass Moment of Inertia

In the following example you build and solve a 2D beam model using the In-Plane Euler Beam application mode. This model describes the eigenfrequency analysis of a simple geometry. A point mass and point mass moment of inertia are used in the model. The two first eigenfrequencies are compared with the values given by an analytical expression.

Model Definition

The geometry consists of a frame with one horizontal and one vertical member. The cross section of both members has an area, A, and an area moment of inertia, I. The length of each member is L and Young's modulus is E. A point mass m is added at the middle of the horizontal member and a point mass moment of inertia J at the corner (see the figure below).



GEOMETRY

- Framework member lengths, $L = 1$ m
- The framework members has a square cross section with a side length of 0.03 m giving an area of $A = 9 \cdot 10^{-4}$ m^2 and an area moment of inertia of $I = 0.03^4/12$ m^4

MATERIAL

Young's modulus, $E = 200$ GPa.

MASS

- Point mass $m = 1000$ kg.
- Point mass moment of inertia $J = mr^2$ where r is chosen to $L/4$.

CONSTRAINTS

The beam is pinned at $x = 0, y = 0$ and $x = 1, y = 1$, meaning that the displacement are constrained whereas the rotational degrees of freedom are free.

Results and Discussion

The analytical values for the two first eigenfrequencies f_{e1} and f_{e2} are given by:

$$\omega_{e1}^2 = \frac{48 \cdot E \cdot I}{m \cdot L^3}$$

$$\omega_{e2}^2 = \frac{48 \cdot 32 \cdot E \cdot I}{7 \cdot m \cdot L^3}$$

and

$$f_{e1} = \frac{\omega_{e1}}{2\pi}$$

$$f_{e2} = \frac{\omega_{e2}}{2\pi}$$

where ω is the angular frequency.

The following table shows a comparison between the eigenfrequencies calculated with COMSOL Multiphysics and the analytical values.

EIGENMODE	COMSOL MULTIPHYSICS	ANALYTICAL
1	4.05 Hz	4.05 Hz
2	8.65 Hz	8.66 Hz

COMSOL plots visualizing the two eigenmodes are shown below.

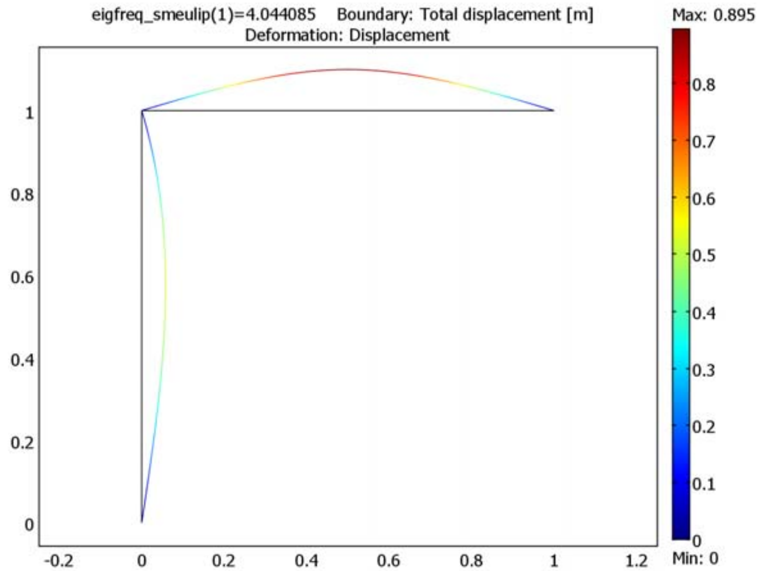


Figure 4-3: First eigenmode.

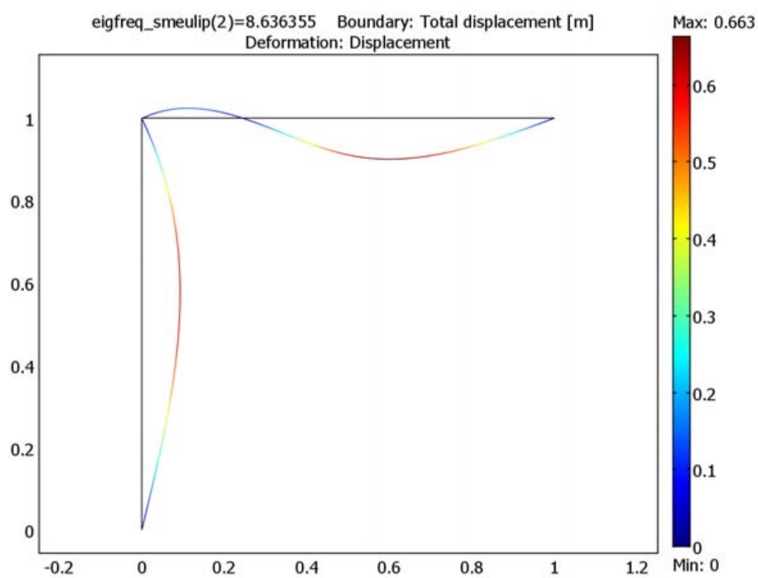


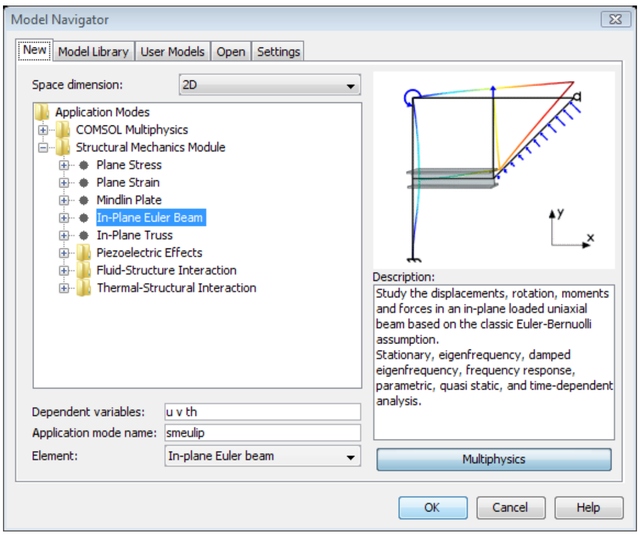
Figure 4-4: Second eigenmode.

Model Library path:

`Structural_Mechanics_Module/Benchmark_Models/in-plane_framework`

MODEL NAVIGATOR

1. Select **2D** in the **Space dimension** list on the **New** page in the **Model Navigator**.
2. Select **Structural Mechanics Module>In-Plane Euler Beam** and click **OK.**

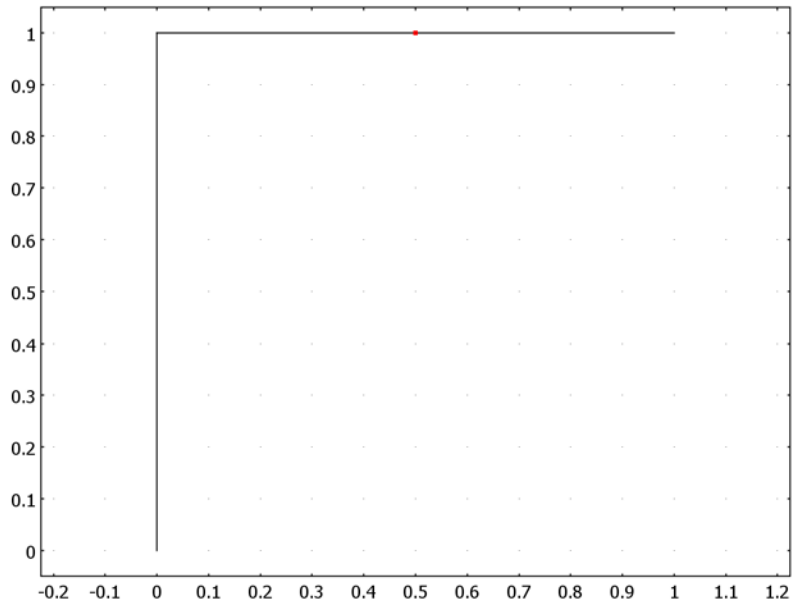


GEOMETRY MODELING

Two lines and an extra point at the middle of the horizontal member is drawn.

1. Click the **Line** button on the Draw toolbar.
2. Draw a line from $x = 0, y = 0$ to $x = 0, y = 1$ by clicking on the left mouse button at these coordinates. End the line by clicking on the right mouse button.
3. Draw a second line from $x = 0, y = 1$ to $x = 1, y = 1$.
4. Click the **Point** button on the Draw toolbar and place a point at $x = 0.5, y = 1$ (to do so, you may have to change the grid spacing to 0.1).

.



OPTIONS AND SETTINGS

1 From the **Options** menu. choose **Constants**.

2 Enter the following constant names, expressions, and descriptions (optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
E	2e11[Pa]	Young's modulus
I	(0.03[m])^4/12	Area moment of inertia
L	1[m]	Framework member length
m	1000[kg]	Point mass
r	L/4	Point mass radius
J	m*r^2	Point mass moment of inertia
A	0.03[m]*0.03[m]	Cross-sectional area
w1	sqrt(48*E*I/(m*L^3))	Angular frequency, eigenfrequency 1
w2	sqrt(48*32*E*I/(7*m*L^3))	Angular frequency, eigenfrequency 2
f1	w1/(2*pi)	Eigenfrequency 1
f2	w2/(2*pi)	Eigenfrequency 2

PHYSICS SETTINGS

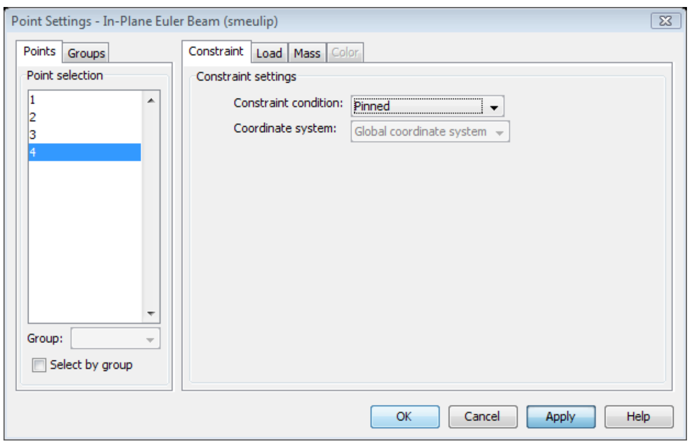
Point Settings

Constrain the x and y-displacements at the beam ends.

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

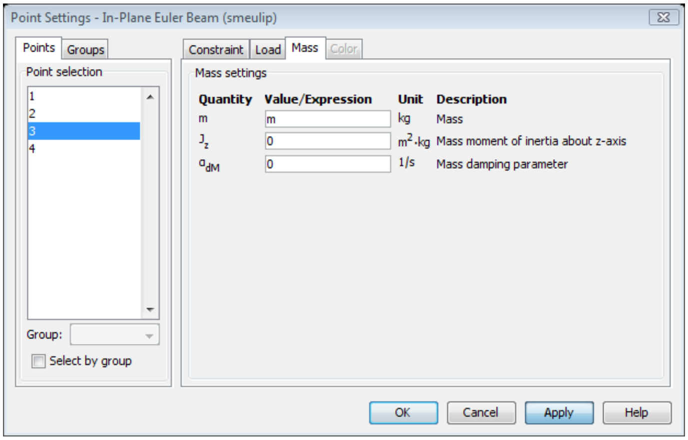
2 Specify constraints on the **Constraint** page according to the following table:

	POINTS 1, 4	
Page	Constraint	
	Constraint condition	Pinned



3 Specify the following mass properties on the **Mass** page:

SETTINGS	POINT 2	POINTS 2, 3
Page	Mass	Mass
Quantity	`mass`	Mass moment of inertia about the z-axis
Expression	`m`	`J`



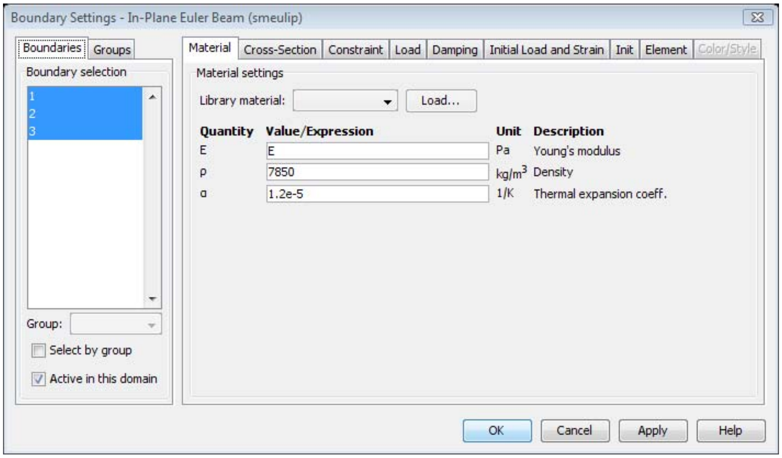
Boundary Settings

Specify the material and cross section properties of the framework members

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

2 Specify boundary settings according to the following table:

	BOUNDARIES 1–3	
Page	Material	
	E	E
Page	Cross Section	
	A	A
	I_{yy}	I



MESH GENERATION

Click the **Initialize Mesh** button on the Main toolbar to mesh the geometry.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 Select **Eigenfrequency** from the **Analysis** list.

3 Enter 2 in the **Desired number of eigenfrequencies** edit field; then click **OK**.

4 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

Plot the two first eigenmodes:

1 Select **Plot Parameters** from the **Postprocessing** menu and go to **General** page.

2 Clear the **Surface** check box and select the **Boundary** and **Deformed shape** check boxes in the **Plot type** area.

3 Select the first eigenmode/eigenfrequency from **Eigenfrequency** list.

4 Click the **Boundary** tab.

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

6 Click **Apply** to view the plot (see Figure 4-3).

7 Select the second eigenmode/eigenfrequency from the **Eigenfrequency** list on the **General** page and click **OK** (see Figure 4-4).

3D Thermally Loaded Beam

In the following example you will build and solve a 3D beam model using the 3D Euler Beam application mode. This model shows how a thermally induced deformation of a beam is modeled. Temperature differences are applied across the top and bottom surfaces as well as the left and right surfaces of the beam. The deformation is compared with the value given by a theoretical solution given in Ref. 1.

Model Definition

GEOMETRY

The geometry consists of one beam. The beam cross-section area is A and the area moment of inertia I. The beam is L long, and the Young's modulus is E.

- Beam length L = 3 m
- The beam has a square cross section with a side length of 0.04 m giving an area of $A = 1.6{\cdot}10^{-3}$ m^2 and an area moment of inertia of $I = 0.04^4/12$ m^4

MATERIAL

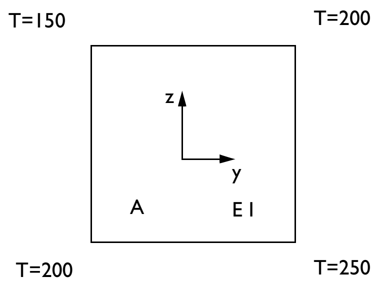
- Young's modulus set to E = 210 GPa
- Poisson's ratio $\nu = 0.3$
- Thermal expansion coefficient $\alpha = 11{\cdot}10^{-6}$/°C

CONSTRAINTS

- Displacements in x, y, and z direction are constrained to zero at x=0, y=0, and z=0.
- Rotation around x-axis are constrained to zero at $x = 0$, $y = 0$, and $z = 0$ to prevent singular rotational degrees of freedom.
- Displacements in the x and y direction are constrained to zero at $x = 3$, $y = 0$, and $z = 0$.

THERMAL LOAD

The surface temperature at each corner of the cross section is depicted in the figure below. The temperature varies linearly between each corner. The deformation caused by this temperature distribution is modeled by specifying the temperature differences across the beam in the local y and z directions.



Results and Discussion

Based on Ref. 1, you can compare the maximum deformation in the global z direction with analytical values for a simply supported 2D beam with a temperature difference between the top and the bottom surface. The maximum deformation (according to Ref. 1) is:

$$w = \frac{\alpha \cdot L^2}{8 \cdot t} \cdot (T_2 - T_1)$$

where t is the depth of the beam (0.04 m), T_2 is the temperature at the top and T_1 at the bottom.

The following table shows a comparison of the maximum global z-displacement, calculated with COMSOL Multiphysics, with the theoretical solution.

W	COMSOL MULTIPHYSICS (MAX)	ANALYTICAL
	15.5 mm	15.5 mm

Figure 4-5 shows the global z-displacement along the beam.

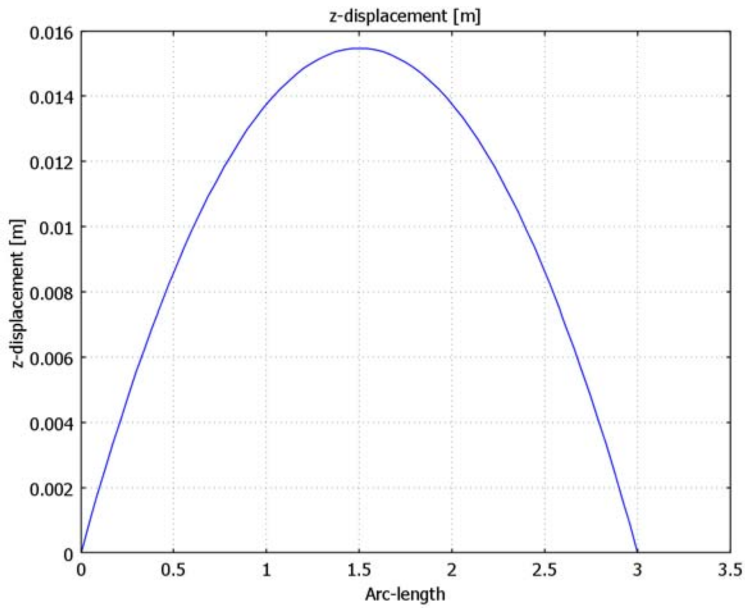


Figure 4-5: z -displacement along the beam

The analytical values for the maximum total camber can be calculated by:

$$\delta = \sqrt{w^2 + v^2}$$

where v is the maximum deformation in the global y direction which is calculated in the same way as w.

A comparison of the camber calculated with COMSOL Multiphysics and the analytical values are shown in the table below.

TOTAL CAMBER	COMSOL MULTIPHYSICS	ANALYTICAL
	22 mm	21.9 mm

Figure 4-6 shows the total camber along the beam.

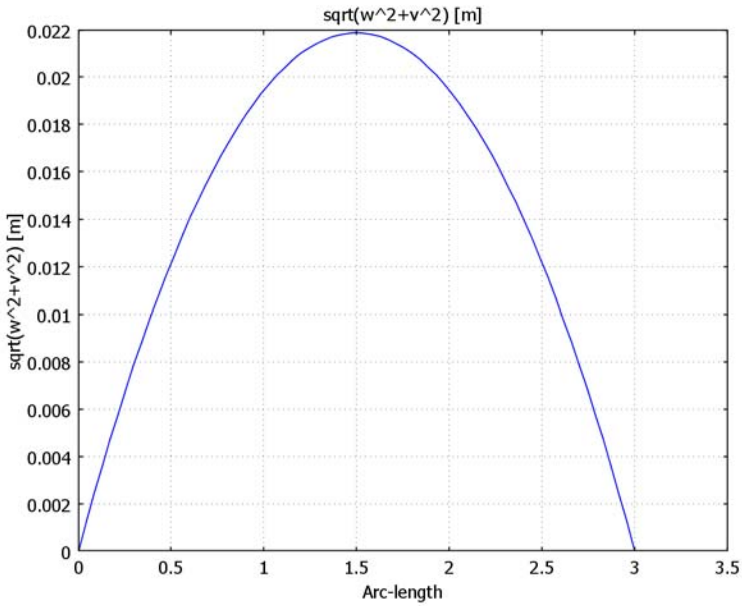


Figure 4-6: Camber along the beam

Reference

1. Young, W, *Roark's Formulas for Stress & Strain*, McGraw-Hill, 1989.

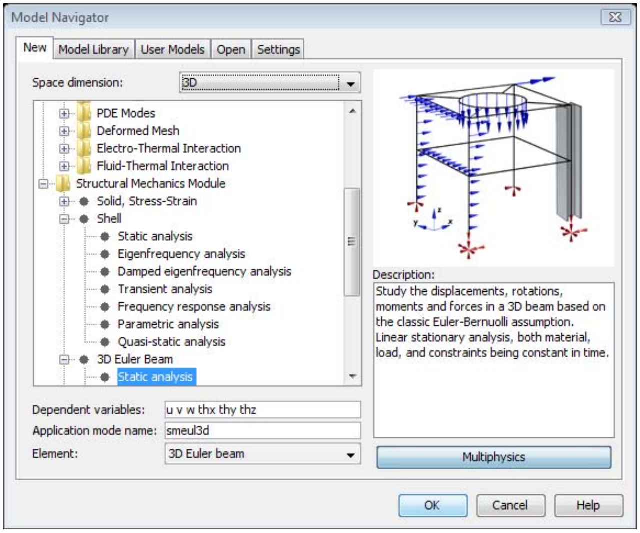
Model Library path: `Structural_Mechanics_Module/Benchmark_Models/thermally_loaded_beam`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **3D** in the **Space dimension** list on the **New** page in the **Model Navigator.**

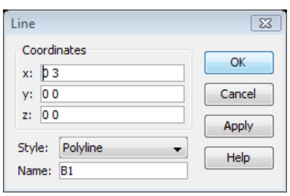
2 Select **Structural Mechanics Module>3D Euler Beam>Static analysis** and click **OK.**

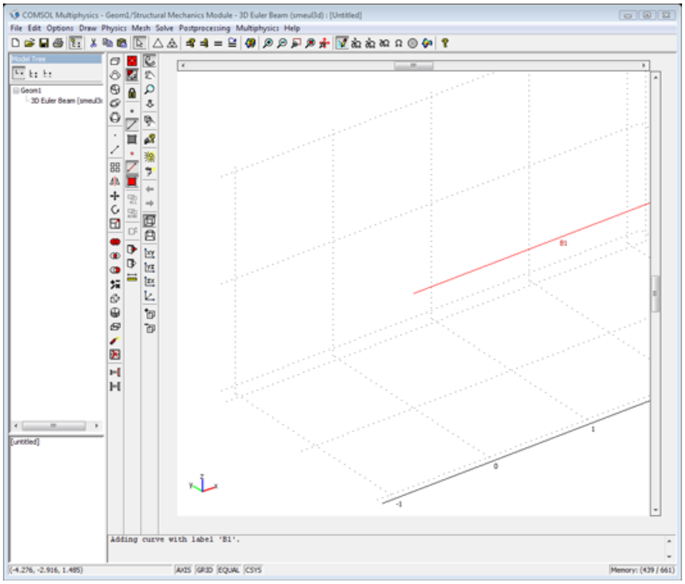


GEOMETRY MODELING

1 Select **Line** from the **Draw** menu.

2 Enter line coordinates according to the figure below and click **OK**.





PHYSICS SETTINGS

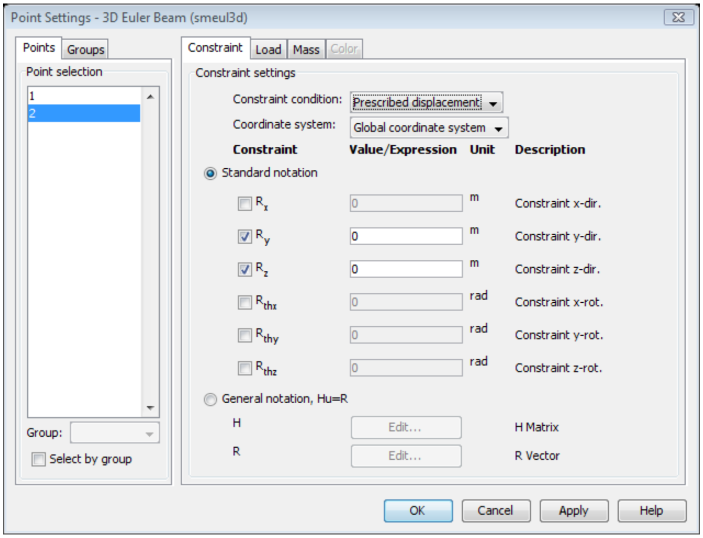
Point Settings

Constrain the *x*-, *y*-, and *z*-displacements at the beam ends.

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

2 Specify constraints on the **Constraint** page according to the following table:

	POINT 1		POINT 2	
Page	Constraint		Constraint	
Constraint condition	Prescribed displacement		Prescribed displacement	
	R_x	0		
	R_y	0	R_y	0
	R_z	0	R_z	0
	R_{thx}	0		



Edge Settings

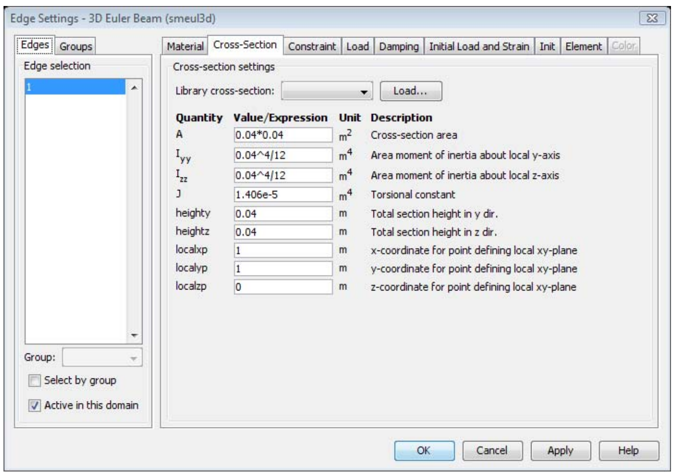
Specify the material and cross section properties of the beam.

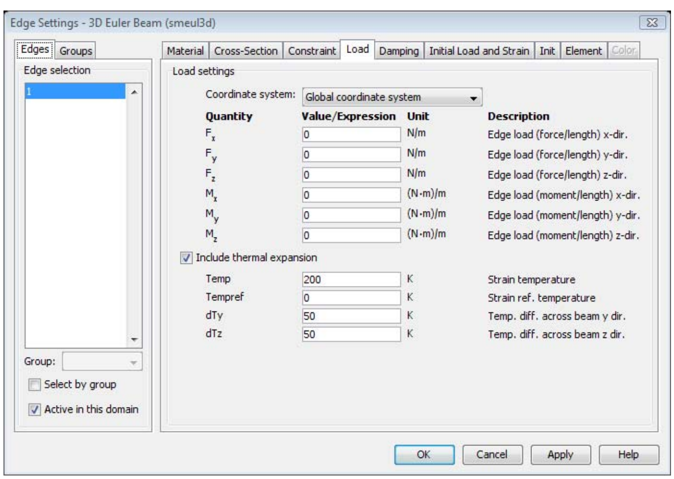
1 Select **Edge Settings** from the **Physics** menu.

2 Specify edge settings according to the following table:

	EDGE 1	
Page	Material	
	Quantity	Expression
	E	210e9
	ν	0.3
	α	11e-6
Page	Cross Section	
	Quantity	Expression
	A	0.04*0.04
	I_{yy}	0.04^4/12
	I_{zz}	0.04^4/12
	heighty	0.04
	heightz	0.04

	EDGE 1	
Page	Load	
	Include thermal expansion	√
	Temp	200
	Tempref	0
	dTy	50
	dTz	50





MESH GENERATION

Use the default mesh settings.

Click the **Initialize Mesh** toolbar button to mesh the geometry.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

Plot the global displacement in the z direction and the total displacement using a domain plot.

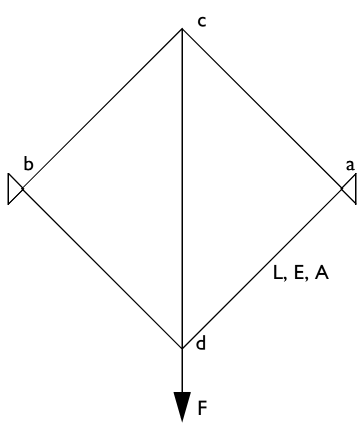
1 Select **Domain Plot Parameters** from the **Postprocessing** menu.
2 Click the **Line/Extrusion** tab.
3 Select Edge `1` from the **Edge selection** list and select **z-displacement** from the **Predefined quantities** list.
4 Click **Apply**; see Figure 4-5.
5 Enter `sqrt(w^2+v^2)` in the **Expression** edit field and click **OK** (see Figure 4-6).

In-Plane Truss

In the following example you build and solve a simple 2D truss model using the In-Plane Truss application mode. This model calculates the deformation of a simple geometry. The example is based on problem 11.1 in *Aircraft Structures for Engineering Students* by T.H.G Megson (Ref. 1). The results are compared with the analytical results given in Ref. 1.

Model Definition

The geometry consists of a square symmetrical truss built up by five members. All trusses have the same cross-sectional area A. The side length is L and the Young's modulus is E.



GEOMETRY

- Truss side length, L = 2 m
- The truss members have a circular cross section with a radius of 0.05 m

MATERIAL

Aluminum: Young's modulus, E = 70 GPa

CONSTRAINTS

Displacements in both directions are constrained at a and b.

LOAD

A vertical force F of 50 kN is applied at the bottom corner.

Results and Discussion

The following table shows a comparison between the results calculated with the Structural Mechanics Module and the analytical results from Ref. 1.

RESULT	COMSOL MULTIPHYSICS	REF. 1
Displacement at d	-5.15E-4 m	-5.15E-4 m
Displacement at c	-2.13E-4 m	-2.13E-4 m
Axial force in member ac=bc	-10.4 kN	-10.4 kN
Axial force in member ad=bd	25.0 kN	25.0 kN
Axial force in member cd	14.6 kN	14.6 kN

The results are in total agreement.

Figure 4-3 shows a plot visualizing the deformed geometry together with the axial forces in the trusses.

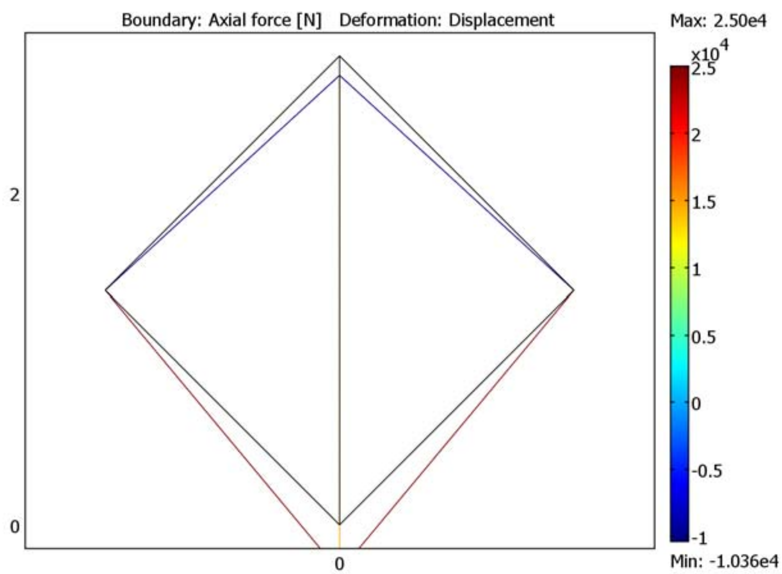


Figure 4-7: Deformed geometry and axial forces.

Reference

1. T.H.G Megson, *Aircraft Structures for Engineering Students*, Edward Arnold, 1985, p. 404.

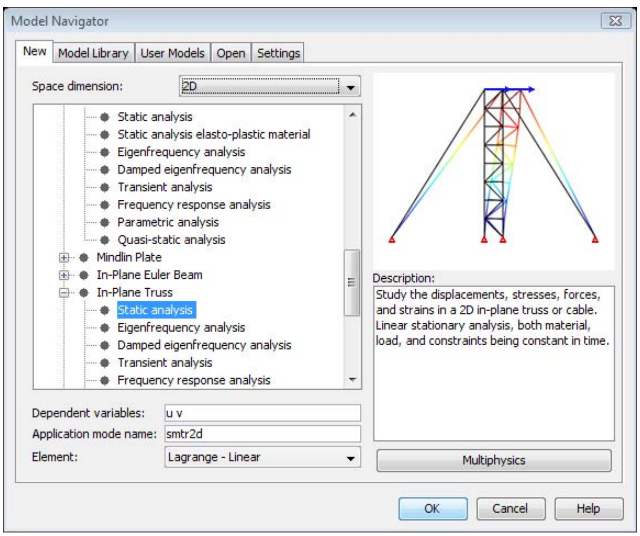
Model Library path:
`Structural_Mechanics_Module/Benchmark_Models/in-plane_truss`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **2D** in the **Space dimension** list on the **New** page in the **Model Navigator.**

2 Select **Structural Mechanics Module>In-Plane Truss** and click **OK**.



OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Axes/Grid Settings**.

2 Set axis and grid settings according to the following table (clear the **Auto** check box before entering the grid spacing):

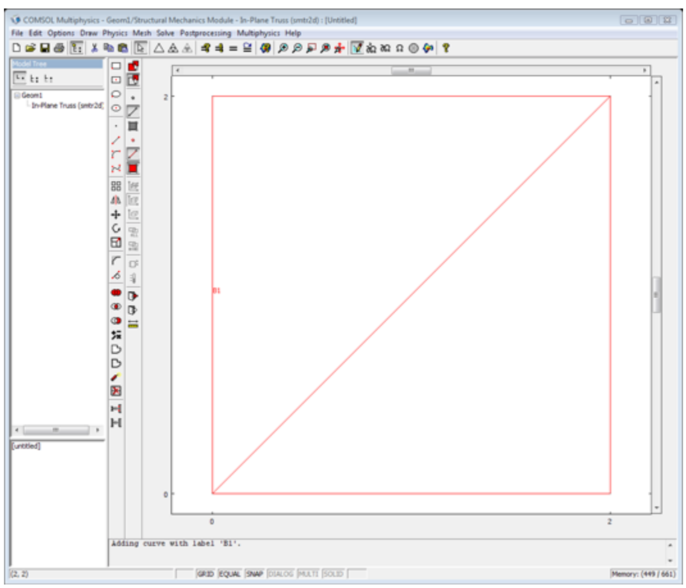
AXIS		GRID	
x min	-4	x spacing	2
x max	4	Extra x	-
y min	-3	y spacing	2
y max	3	Extra y	-

GEOMETRY MODELING

1 Click the **Line** button on the Draw toolbar.

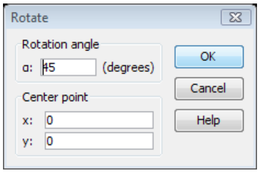
2 Double-click the **SOLID** button in the Status bar, to turn off the solid feature when drawing the lines.

3 Draw a line from (0, 0) through (0, 2), (2, 2), (2, 0), (0, 0), and (2, 2) by clicking on the left mouse button at these coordinates. End the line by clicking on the right mouse button.

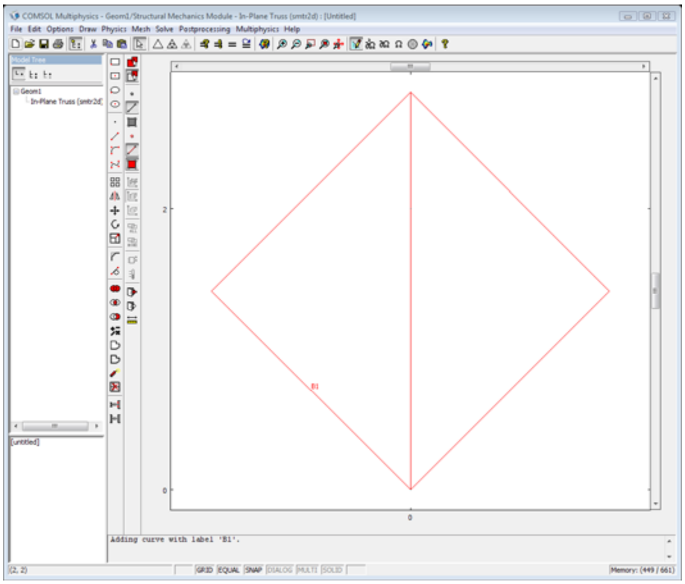


4 Click the **Rotate** button on the Draw toolbar to open the **Rotate** dialog box.

5 Enter 45 as the rotating angle and click **OK** to rotate the truss.



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



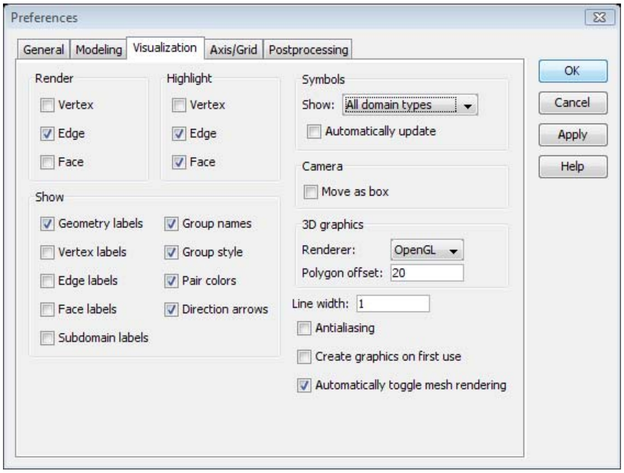
PHYSICS SETTINGS

One helpful feature when specifying loads and constraints are symbols. Use this feature to see where you have specified constraints and applied forces.

1 Select **Preferences** from the **Options** menu to open the **Preferences** dialog box.

2 Click the **Visualization** tab, and select **All domain types** from the **Show** list in the **Symbols** area.

3 Select the **Automatically update** check box and click **OK**.



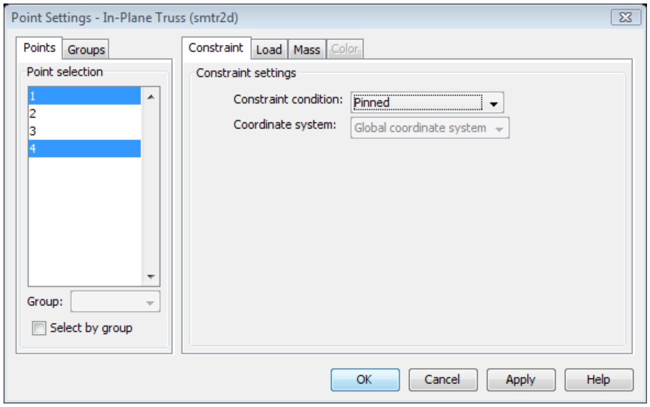
Point Settings

Constrain the x- and y-displacements at the left and right corners of the truss.

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

2 Specify constraints on the **Constraint** page according to the following table:

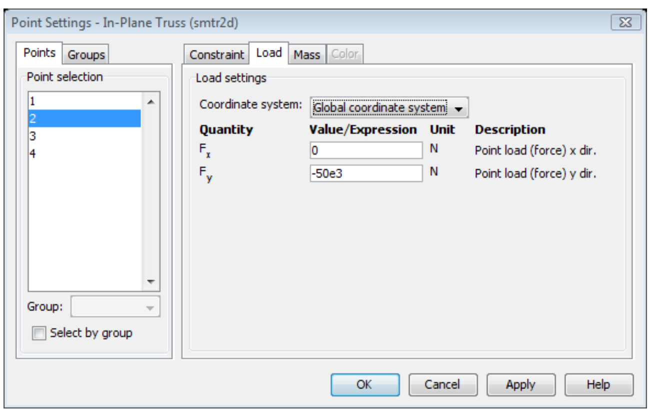
	POINTS 1, 4	
Page	Constraint condition	
	Pinned	



Specify the vertical force at the bottom corner:

1 Click the **Load** tab in the **Point Settings** dialog box.

2 Select Point 2 and enter `-50e3` in the $\mathbf{F_y}$ edit field.



3 Click **OK** to close the **Point Settings** dialog box.

Boundary Settings

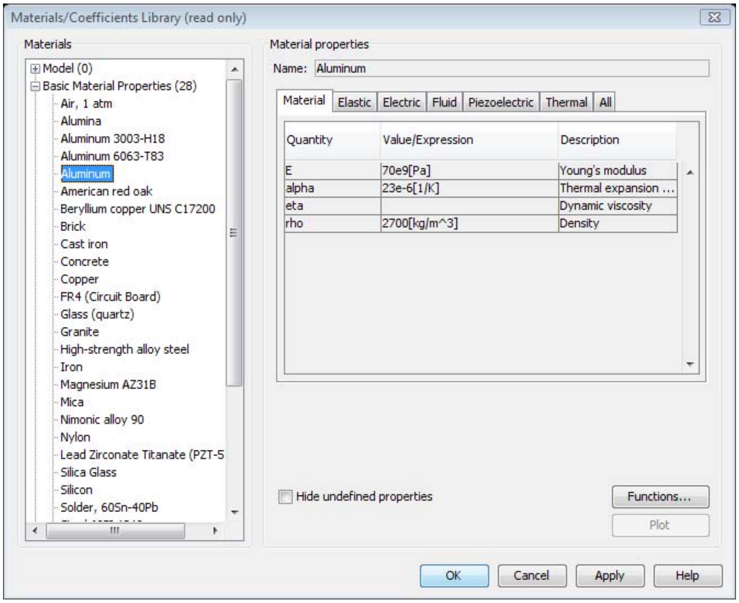
Specify the material and cross-section properties of the truss members:

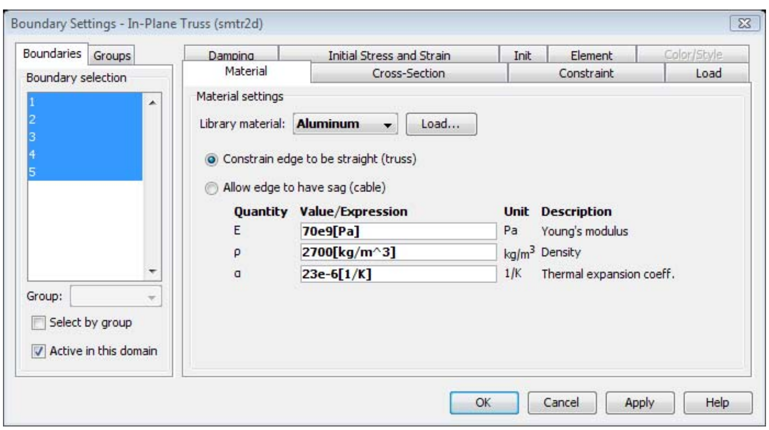
1 Select **Boundary Settings** from the **Physics** menu.

2 Select all five boundaries.

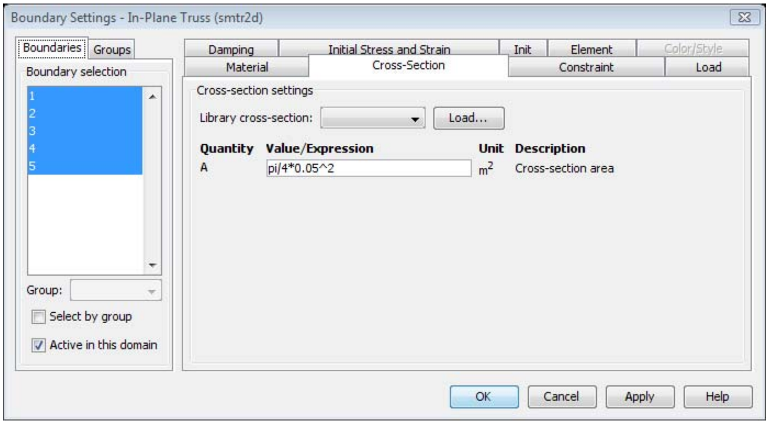
3 Click the **Load** button on the **Material** page to open the **Material/Coefficients Library** dialog box.

4 Select **Aluminum** from **Basic Material Properties** in the **Materials** list and click **OK**.

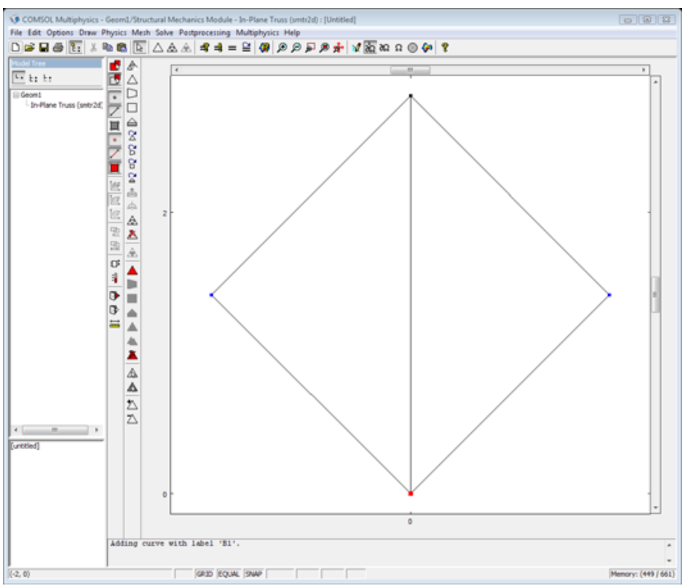




5 Click the **Cross Section** tab, and enter `pi/4*0.05^2` in the **Cross section area** (**A**) edit field.



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



MESH GENERATION

When using the default option (the **Constrain edge to be straight (truss)** button in selected on the **Material** tab in the **Boundary Settings** dialog box) the mesh is not critical. The **Allow edge to have sag (cable)** option makes the mesh critical. The reason for this is that the internal nodes along the boundary become singular because they do not have any stiffness perpendicular to the boundary. You can solve this by using a very coarse mesh with no internal nodes along the boundaries. For more information, see "Straight Edge Option" on page 281 of the *Structural Mechanics Module User's Guide*.

In this model, use the default mesh settings, as the model uses the default option for a straight edge (the **Constrain edge to be straight (truss)** button is selected on the **Material** page in the **Boundary Settings** dialog box).

Click the **Initialize Mesh** toolbar button to mesh the geometry.

COMPUTING THE SOLUTION

Click the **=** toolbar button to start the analysis and compute the solution.

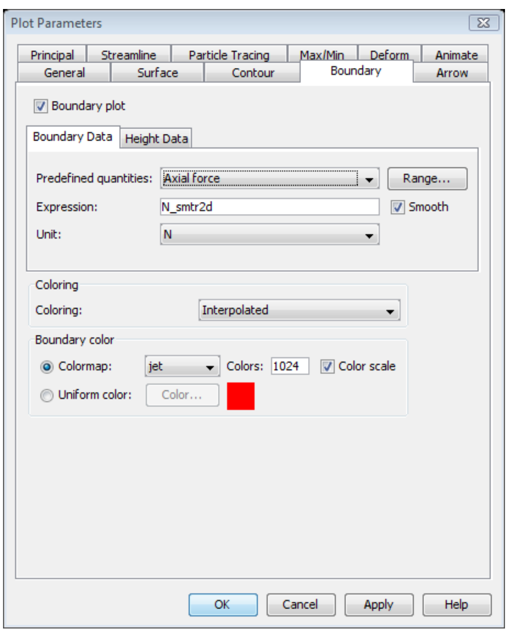
POSTPROCESSING AND VISUALIZATION

Plot the deformed geometry together with the axial forces in the truss members.

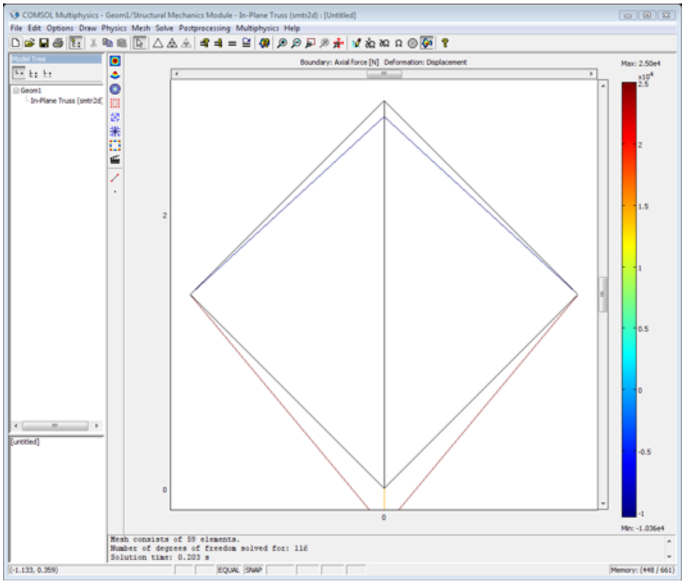
1 Select **Plot Parameters** from the **Postprocessing** menu.

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

3 Click the **Boundary** tab, and select **Axial force** from the **Predefined quantities** list.



4 Click **OK** to close the dialog box and view the plot.



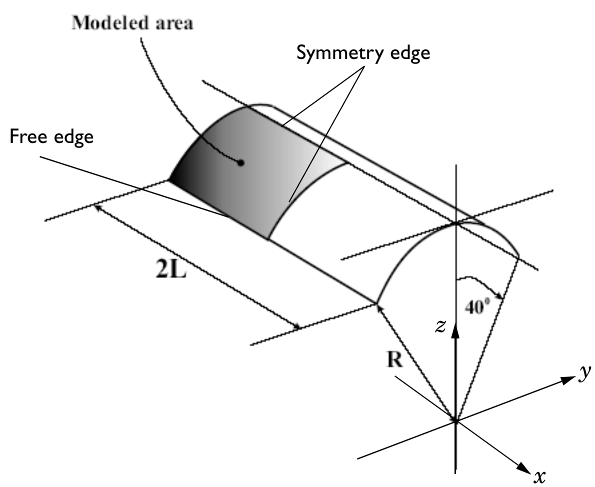
Scordelis-Lo Roof Shell Benchmark

In the following example you build and solve a 3D shell model using the Shell application mode. This model is a widely used benchmark model denoted the Scordelis-Lo roof. The computed maximum z-deformation is compared with the value given in Ref. 1.

Model Definition

GEOMETRY

The geometry consists of a curved face as depicted in the figure below. Only one quarter is analyzed due to symmetry.



- Roof length $2L = 50$
- Roof radius $R = 25$.

MATERIAL

- Isotropic material with Young's modulus set to $E = 4.32 \cdot 10^8$.
- Poisson's ratio $\nu = 0.0$.

CONSTRAINTS

- The outer straight edge is free.
- The outer curved edge of the model geometry is constrained in the y and z directions.
- The straight symmetry edge on the top of the roof has symmetry edge constraints, that is, translation in the y direction is constrained as well as the rotations about the x-axis and z-axis.
- The curved symmetry edge has also symmetry constraints, that is, translation in the x direction is constrained as well as the rotations about the y-axis and z-axis.

LOAD

A force per area unit of -90 N/m^2 in the z direction is applied on the surface.

Results and Discussion

The maximum deformation in the global z direction with the default mesh settings is depicted in Figure 4-8.

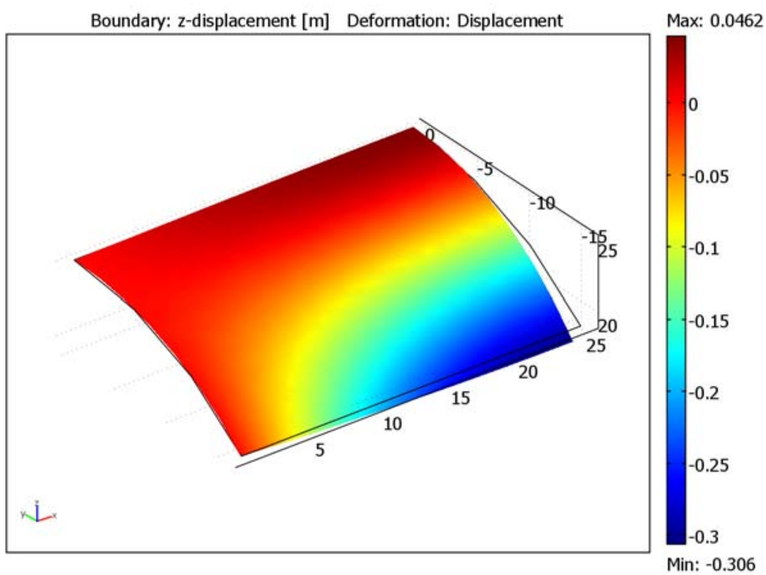


Figure 4-8: z-displacement with 242 elements.

The reference solution quoted in Ref. 1 for the midside vertical displacement is 0.3086 m. The FEM solution converges toward 0.302 m with a refined mesh; see Figure 4-9 and Figure 4-10.

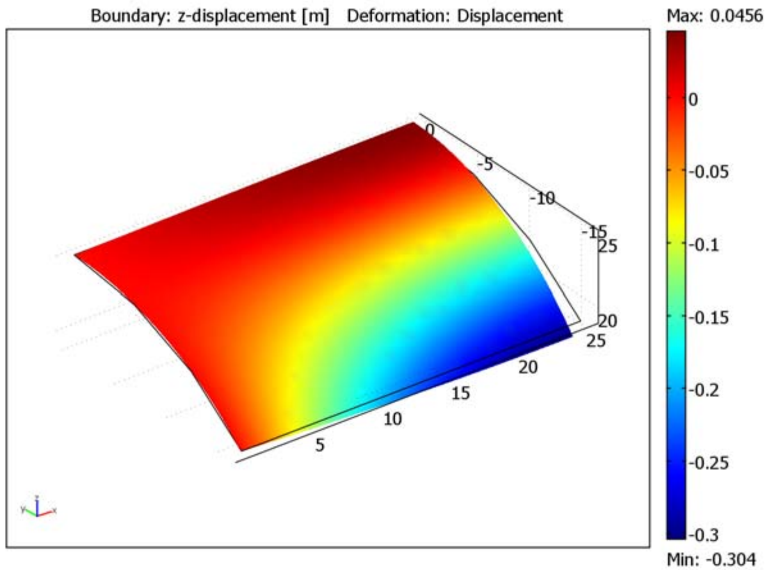


Figure 4-9: z-displacement with 832 elements.

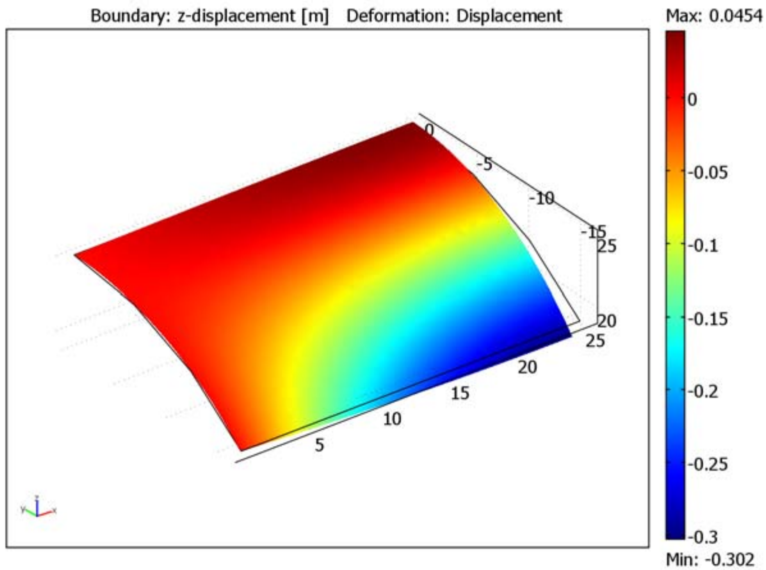


Figure 4-10: z-displacement with 1942 elements

This value (0.302) is in fact observed in other published benchmarks treating this problem as the value which this problem converges toward.

Reference

1. MacNeal R.H., Harder R.L., *Proposed Standard Set of Problems to Test Finite Element Accuracy*, Finite Elements in Analysis and Design, 1, 1985.

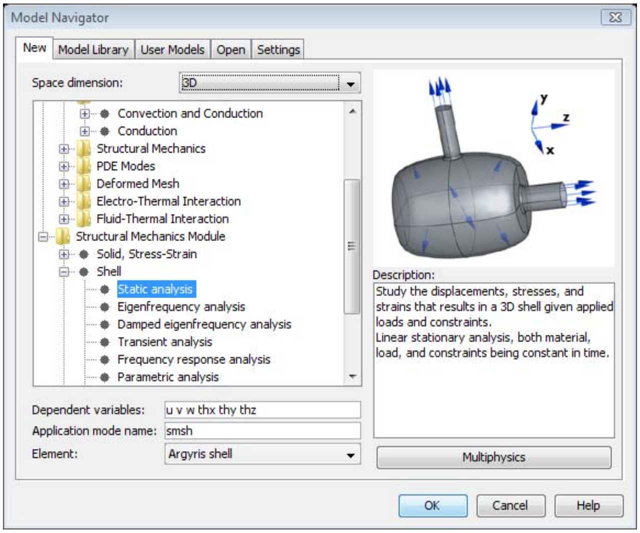
Modeling Using the Graphical User Interface

Model Library path: `Structural_Mechanics_Module/Benchmark_Models/scordelis_lo_roof`

MODEL NAVIGATOR

1 Select **3D** in the **Space dimension** list on the **New** page in the **Model Navigator**.

2 Select **Structural Mechanics Module>Shell>Static analysis** and click **OK.**



GEOMETRY MODELING

1 Select **Work-Plane Settings** from the **Draw** menu.

2 Click **OK** in the **Work-Plane Settings** dialog box. This gives you the default work plane.

3 Click the **Line** button and draw a line from (0, 25) to (25, 25).

4 Select **Revolve** from the **Draw** menu.

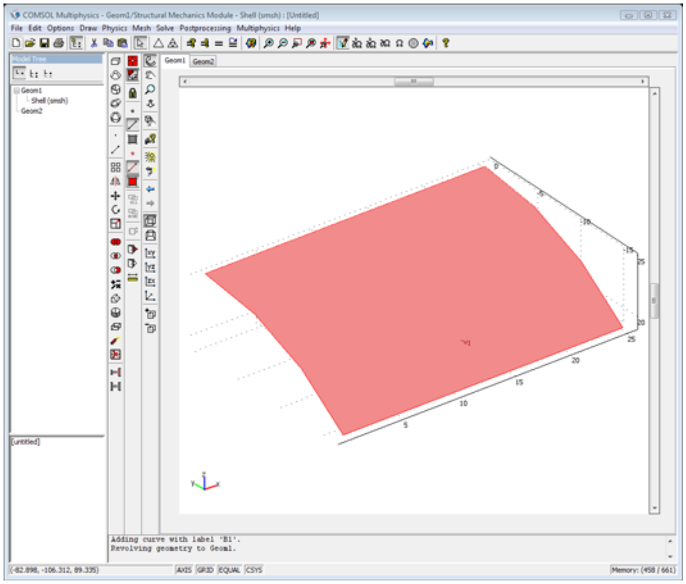
5 Type `90` in the **α1** edit field.

6 Type `90+40` in the **α2** edit field.

7 Type `1` in the **Second point x** edit field.

8 Type `0` in the **Second point y** edit field

9 Click **OK**.



PHYSICS SETTINGS

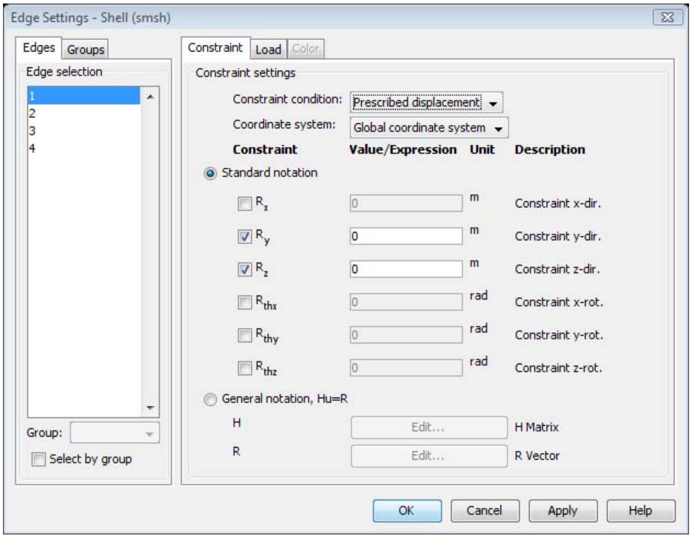
Edge Settings

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

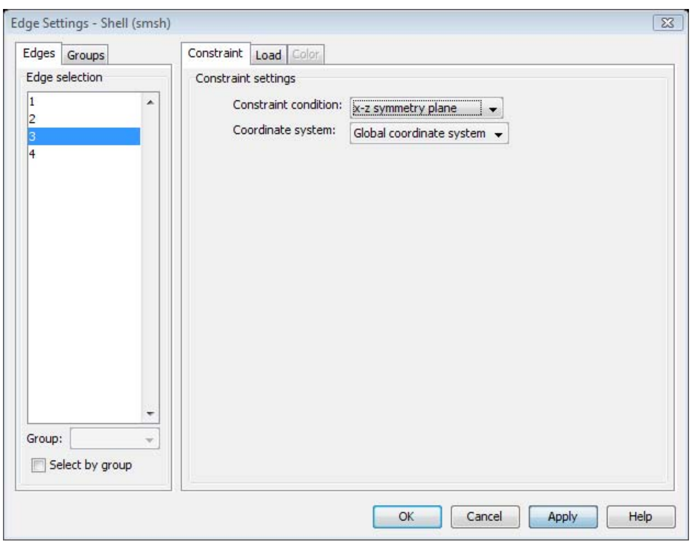
2 Specify constraints on the **Constraint** page according to the following table:

	EDGE 1		EDGE 3		EDGE 4	
Page	Constraint		Constraint		Constraint	
Constraint condition	Prescribed displacement		x-z symmetry plane		y-z symmetry plane	
	R_y	0				
	R_z	0				

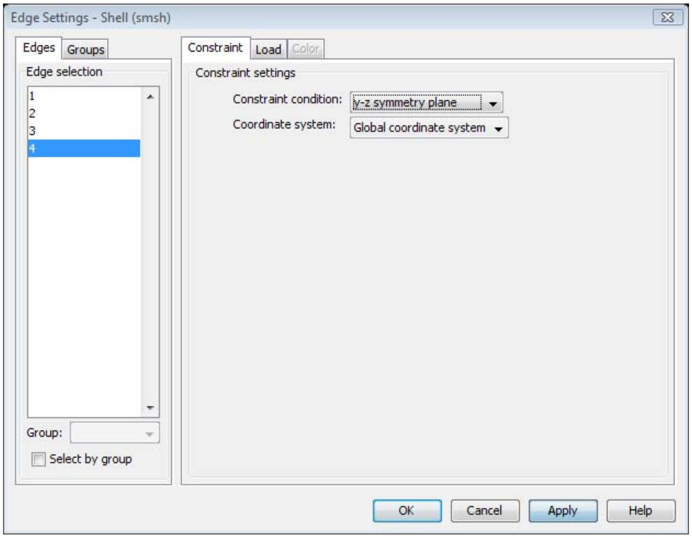
Constraint settings for Edge 1:



Constraint settings for Edge 3:



Constraint settings for edge 4:



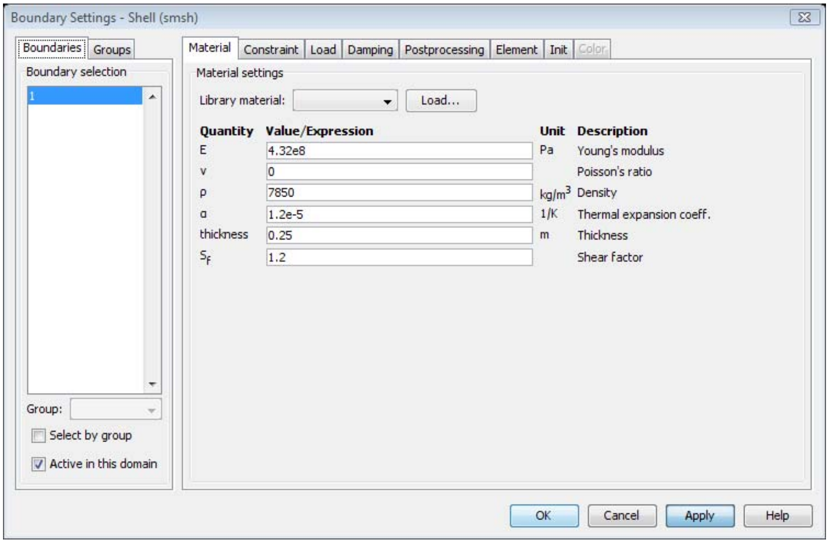
Boundary Settings

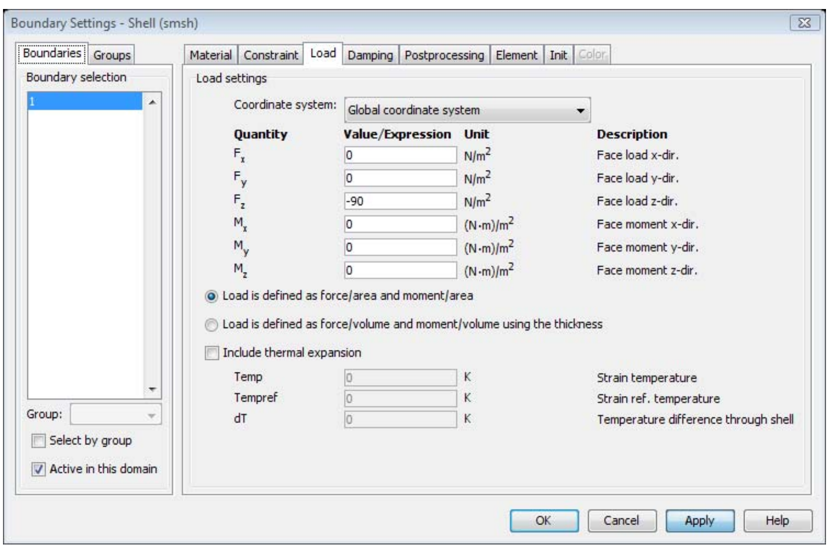
Specify the material properties and surface load in the **Boundary Settings** dialog box.

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

2 Specify boundary settings according to the following table:

	BOUNDARY 1	
Page	Material	
	E	4.32e8
	ν	0.0
	thickness	0.25
Page	Load	
	F_z	-90





MESH GENERATION

Use the default mesh settings. Click the **Initialize Mesh** toolbar button to mesh the geometry.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

Plot the global displacement in the *z* direction and the deformed shape.

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

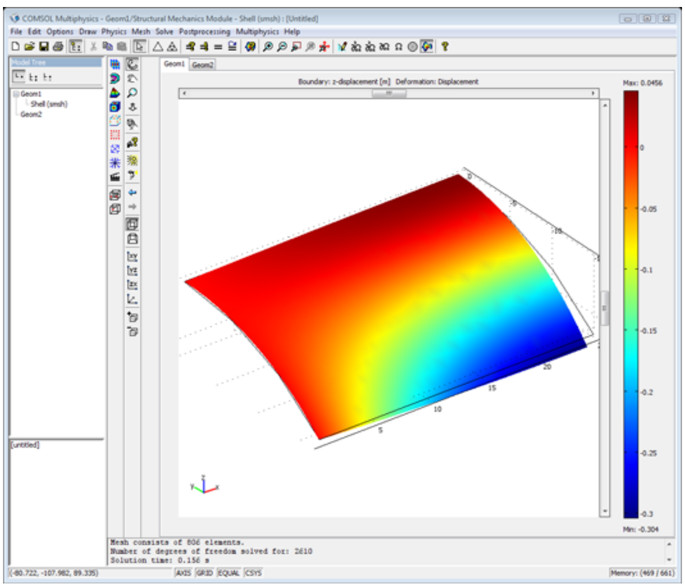
2 Clear the **Slice** check box in the **Plot type** area on the **General** page.

3 Select the **Deformed shape** and **Boundary** check boxes in the **Plot type** area.

4 Click the **Boundary** tab.

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

6 Click **OK**.



Check the convergence by solving the problem twice with different mesh refinements.

MESH GENERATION

1 Choose **Free Mesh Parameters** from the **Mesh** menu.

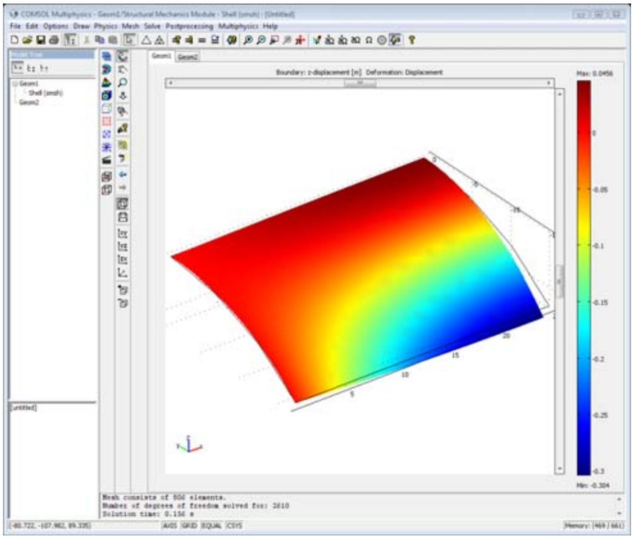
2 Select **Finer** from the **Predefined mesh sizes** list.

3 Click the **Remesh** button.

4 Click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button.

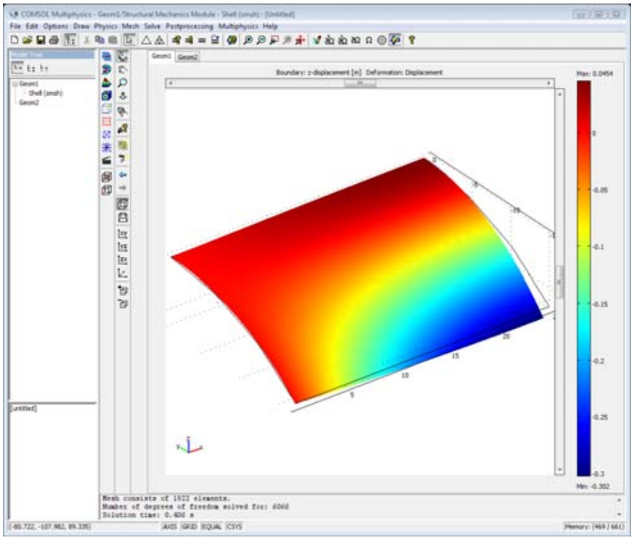


MESH GENERATION

1. Select **Free Mesh Parameters** from the **Mesh** menu.
2. Select **Extra fine** from the **Predefined mesh sizes** list.
3. Click the **Remesh** button.
4. Click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button.



5

Bioengineering

This section contains examples from the bioengineering field.

Fluid-Structure Interaction in a Network of Blood Vessels

Introduction

This model studies a portion of the vascular system, in particular the upper part of the aorta (Figure 5-1). The aorta and its ramified blood vessels are embedded in biological tissue, specifically the cardiac muscle. The flowing blood applies pressure to the artery's internal surfaces and its branches, thereby deforming the tissue. The analysis consists of two distinct but coupled procedures: first, a fluid-dynamics analysis including a calculation of the velocity field and pressure distribution in the blood (variable in time and in space); second, a mechanical analysis of the deformation of the tissue and artery. In this model, any change in the shape of the vessel walls does not influence the fluid domain, which implies that there is only a 1-way fluid-structural coupling. However, in COMSOL Multiphysics it is possible to simulate a 2-way coupling using the ALE (arbitrary Lagrangian-Eulerian) method.

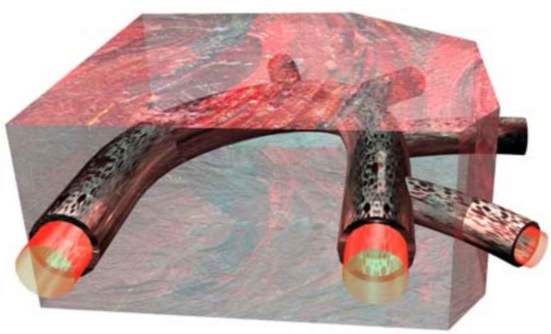


Figure 5-1: The model domain consists of part of the aorta, its branches, and the surrounding tissue.

Model Definition

Figure 5-2 shows two views of the problem domain, one with and one without the cardiac muscle. The model's mechanical analysis must consider the cardiac muscle because it presents a stiffness that resists artery deformation due to the applied pressure.

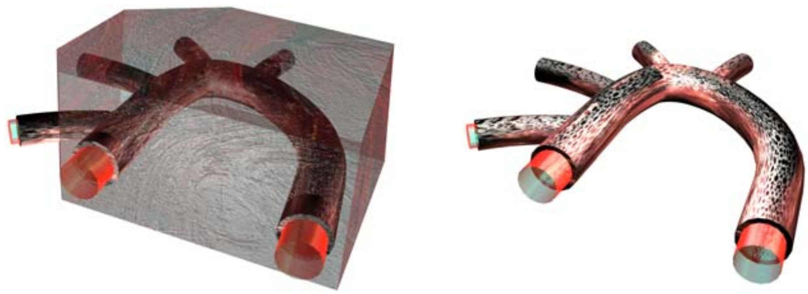


Figure 5-2: A view of the aorta and its ramification (branching vessels) with blood contained, shown both with (left) and without (right) the cardiac muscle.

The main characteristics of the analyses are:

- *Fluid dynamics analysis*

  Here the model solves the Navier-Stokes equations. This analysis considers only the subdomains of the blood. At the point where the model brings the vessels to an abrupt end it represents the load with a known pressure distribution.

- *Mechanical analysis*

  This analysis is *highly nonlinear* due to the assumption of a large displacement and the constitutive behavior of the materials (the model describes the response of the artery and cardiac muscle with an *hyperelastic law* similar to those for rubber components). Only the subdomains related to the biological tissues are active in this analysis. The model represents the load with the pressure distribution it computes during the fluid-dynamics analysis.

ANALYSIS OF RUBBER-LIKE TISSUE AND ARTERY MATERIAL MODELS

The analysis of rubber-like elastomers is generally a difficult task for several reasons:

- The material can undergo very large strains (finite deformations).
- The stress-strain relationship is generally nonlinear.
- Many rubber-like materials are almost incompressible. You must revise standard displacement-based finite element formulations in order to arrive at correct results (mixed formulations).

You must pay particular attention to the definition of stress and strain measures. Finite deformations are displacements where standard assumptions about infinitesimal displacements are no longer valid. It is permissible to consider finite deformations in a model when:

- Significant rigid-body rotations occur (finite rotations).
- The strains are no longer small (larger then a few percent).
- The loading of the body depends on the deformation.

Luckily, all of these issues are dealt with in the hyperelastic material model built-in the Structural Mechanics Module.

MATERIALS

The model in this discussion uses the following material properties:

- Blood
  - density = 1060 kg/m^3
  - dynamic viscosity = 0.005 Ns/m^2
- Artery
  - density = 960 kg/m^3
  - Neo-Hookean hyperelastic behavior: the coefficient μ equals 6,204,106 N/m^2, while the bulk modulus equals 20 μ and corresponds to a value for Poisson's ratio, ν, of 0.45. An equivalent elastic modulus equals $1.0 \cdot 10^7$ N/m^2.
- Cardiac muscle
  - density = 1200 kg/m^3
  - Neo-Hookean hyperelastic behavior: the coefficient μ equals 719,676 N/m^2, while the bulk modulus equals 20 μ and corresponds to a value for Poisson's ratio, ν, of 0.45. An equivalent elastic modulus equals $1.16 \cdot 10^6$ N/m^2.

FLUID DYNAMICS ANALYSIS

The fluid dynamics analysis considers the solution of the 3D Navier-Stokes equations. You can do so in both a stationary case or in the time domain. To establish the boundary conditions, the model uses six pressure conditions with the configuration in Figure 5-3.

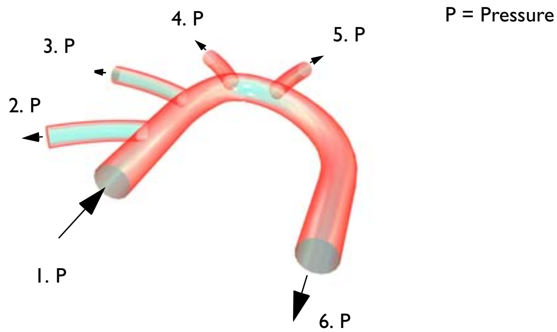


Figure 5-3: Boundary conditions for the fluid-flow analysis.

The pressure conditions are:

- Section 1: $11{,}208\ \mathrm{N/m^2}$
- Section 2: $11{,}192\ \mathrm{N/m^2}$
- Section 3: $11{,}148\ \mathrm{N/m^2}$
- Section 4: $11{,}148\ \mathrm{N/m^2}$
- Section 5: $11{,}148\ \mathrm{N/m^2}$
- Section 6: $11{,}120\ \mathrm{N/m^2}$

For the time-dependent analysis, the model uses a simple trigonometric function to vary the pressure distribution over time:

$$f(t) = \begin{cases} \sin \pi t & 0 \le t \le \frac{1}{2}\mathrm{s} \\ \frac{3}{2} - \frac{1}{2} \cdot \cos\left(2\pi\left(t - \frac{1}{2}\right)\right) & \frac{1}{2}\mathrm{s} \le t \le \frac{3}{2}\mathrm{s} \end{cases} \tag{5-1}$$

You can implement this effect in COMSOL Multiphysics with an expression that you define on the active subdomain.

Results and Discussion

First examine the steady flow field so as to compare its result to the transient case. Figure 5-4 shows a slice plot of the velocity field (that gives values in m/s) and a streamline plot (that shows velocity as a directional vector). The model is of moderate size with a total of 10,000 elements for the fluid-flow analysis.

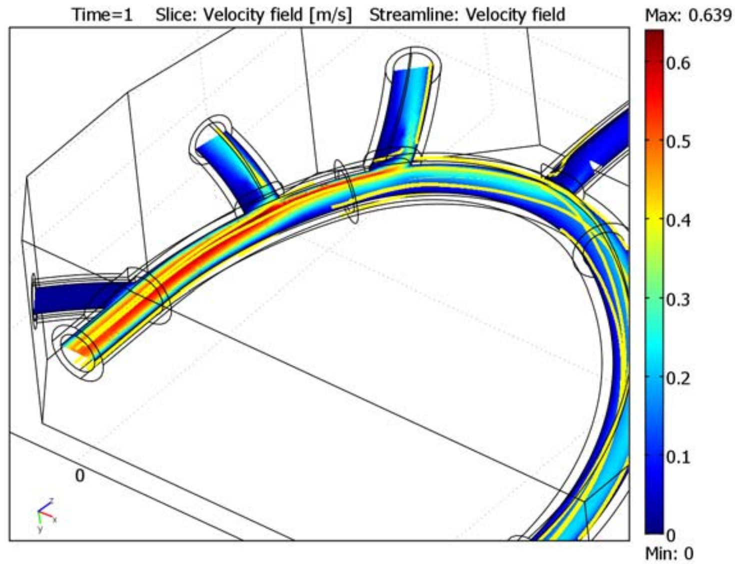


Figure 5-4: Velocity field color slice and flow lines in the aorta and its ramification (branching).

In comparison, the transient analysis reveals only a small difference between two pressure distributions (the one computed with the time-dependent analysis, and the one from the stationary analysis with pressure defined as in Equation 5-1). In other words, the pressure field reaches its steady state very quickly when changing the boundary conditions.

Figure 5-5 shows the pressure at a point in center of the vessel as computed with a time-dependent model where the boundary conditions change at a slow, continuous rate. The small difference implies that you can scale the pressure computed in a fluid-dynamics stationary analysis in the subsequent parametric nonlinear mechanical analysis by multiplying the value *p* with the Amplitude function in Equation 5-1.

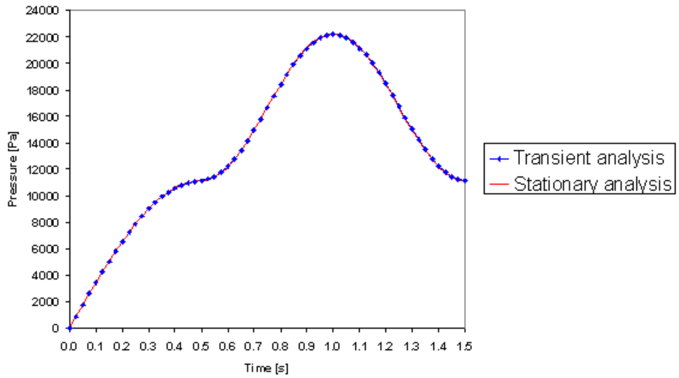


Figure 5-5: Pressure at a given point in the transient model (data points) and the steady-flow model (solid line). Both cases take the value of the boundary condition from Equation 5-1, but the steady-flow model varies time using the parametric solver assuming a steady-state condition at every point.

A final model accounts for the influence of large displacements, and it accounts for the hyperelastic behavior of the biological tissues.

Figure 5-6 shows the total displacement at the peak load (after 1 s).

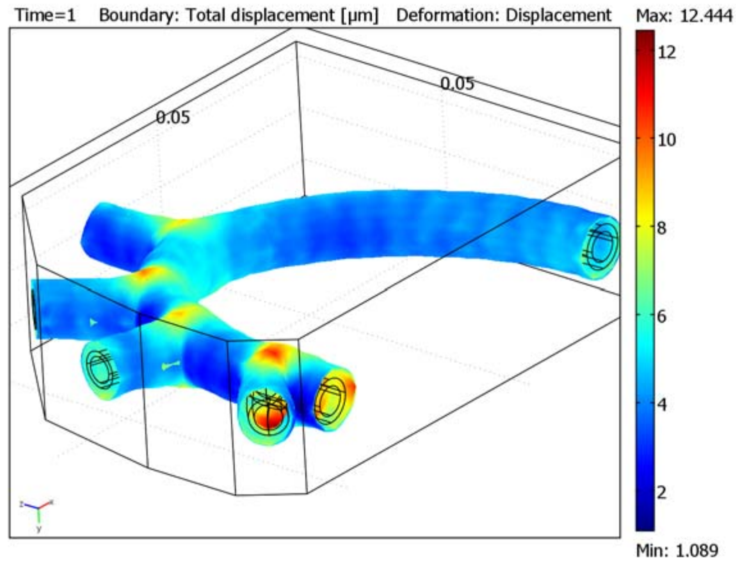


Figure 5-6: Displacements in the blood vessel using a hyperelastic model.

Model Library path: `Structural_Mechanics_Module/Bioengineering/blood_vessel`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Double-click the COMSOL Multiphysics icon on your desktop to open the **Model Navigator**.

2 Select **3D** from the **Space dimension** list.

3 In the list of application modes, select the **COMSOL Multiphysics>Fluid Dynamics>Incompressible Navier-Stokes** application mode.

4 Click **OK**.

You solve the problem using SI units (meters, seconds, newtons, and so on). Because the plane-view geometry is measured in centimeters, first enter the geometry as if the length unit were centimeter and then scale each coordinate direction by 10^{-2} to obtain the geometry in SI units (meters). This approach reduces the amount of typing and reduces the risk of errors when building the geometry.

OPTIONS AND SETTINGS

Constants

1 Select **Constants** from the **Options** menu and enter constants with names, expressions, and descriptions (optional) according to the following table:

NAME	EXPRESSION	DESCRIPTION
p1	11208[Pa]	Pressure condition 1
p2	11192[Pa]	Pressure condition 2
p3	11148[Pa]	Pressure condition 3
p4	11148[Pa]	Pressure condition 4
p5	11148[Pa]	Pressure condition 5
p6	11120[Pa]	Pressure condition 6
rho_blood	1060[kg/m^3]	Density of blood
eta_blood	0.005[Pa*s]	Dynamic viscosity of blood
m_muscle	719676[Pa]	Neo-Hookean hyperelastic behavior, mu coefficient for muscle
v_muscle	20*m_muscle	Bulk modulus for muscle
rho_muscle	1200[kg/m^3]	Density of muscle
m_artery	6204106[Pa]	Neo-Hookean hyperelastic behavior, mu coefficient for artery
v_artery	20*m_artery	Bulk modulus for artery
rho_artery	960[kg/m^3]	Density of artery

2 Click **OK**.

GEOMETRY MODELING

Import the model geometry from a CAD file:

1 Choose **File>Import>CAD Data From File**.

2 In the **Look in** list browse to the folder `models/Structural_Mechanics_Module/Bioengineering` in your COMSOL Multiphysics installation directory.

3 Select the file `blood_vessel.mphbin`, then click **Import**.

Rescaling the Geometry

1 Press Ctrl+A to select all objects.

2 Click the **Scale** button on the Draw toolbar.

3 Type `1e-2` in the **x**, **y**, and **z** edit fields

4 Click **OK**.

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

PHYSICS SETTINGS

Start by defining the physical properties in the different subdomains.

Subdomain Settings

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

2 Select Subdomains 1 and 2 in the **Subdomain Selection** list and clear the **Active in this subdomain** box to deactivate these subdomains.

3 Select Subdomain 3 and type `rho_blood` in the **Density** edit field and type `eta_blood` in the **Dynamic Viscosity** edit field. Note that you have already defined these variables.

4 Click **OK**.

Boundary Conditions

Before defining the boundary conditions, you need to define an expression for the heart frequency.

1 Select **Expressions>Scalar Expressions** in the **Options** menu.

2 Type `amp` in the **Name** column and type `(t<=0.5)*sin(pi*t[1/s])+(t>0.5)*(1.5-0.5*cos(-2*pi*(0.5-t[1/s])))` in the **Expression** column.

   The reason for the unit multiplication with `[1/s]` is to make the input to the trigonometric functions nondimensional.

3 Select **Boundary Settings** in the **Physics** menu.

4 In the **Boundary selection** list, select Boundary 38. Set the **Boundary type** to **Inlet** and the **Boundary condition** to **Pressure, no viscous stress**. In the **Pressure** edit field, type `p1*amp`.

5 Select Boundaries 9, 19, 41, 70, and 86 and set the **Boundary type** to **Outlet**.

6 Select each of the six boundaries in turn and specify the **Pressure** according to the following table:

PROPERTY	BOUNDARY 9	BOUNDARY 19	BOUNDARY 41	BOUNDARY 70	BOUNDARY 86
p_0	p3*amp	p2*amp	p4*amp	p5*amp	p6*amp

For all other boundaries, the default **No slip** boundary condition applies.

7 Click **OK**.

MULTIPHYSICS

The next step is to add the structural analysis to the existing model.

1 Select **Model Navigator** in the **Multiphysics** menu.

2 Select the **Solid, Stress-Strain>Quasi-static analysis** application mode from **Structural Mechanics Module** to the model.

3 Click **Add**, then click **OK**.

PHYSICS SETTINGS

Subdomain Settings

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

2 Select Subdomain 3 in the **Subdomain selection** list and clear the **Active in this subdomain** box.

3 Select Subdomain 1 and select **Hyperelastic material** in the **Material model** list.

4 Type `m_muscle` in the **Initial shear modulus** edit field, `v_muscle` in the **Initial bulk modulus** edit field and `rho_muscle` in the **Density** edit field.

5 Click the **Element** tab and select **Lagrange-linear** in the **Predefined elements** drop-down menu.

6 Click the **Material** tab. Select Subdomain 2 and select **Hyperelastic material** in the **Material model** drop-down list.

7 Type `m_artery` in the **Initial shear modulus** edit field, `v_artery` in the **Initial bulk modulus** edit field and `rho_artery` in the **Density** edit field.

8 Click the **Element** tab and select **Lagrange-linear** in the **Predefined elements** list.

9 Click **OK**.

10 Select **Properties** from the **Physics** menu

11 Select **On** in the **Large deformation** list.

12 Click **OK**.

Boundary Conditions

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

2 Select Boundaries 1–6, 12, 26, 27, 30, 33, 64, 67, 85, and 87.

3 Select **Roller** from the **Constraint condition** list.

4 Select the remaining boundaries: 10, 11, 16, 17, 20, 21, 23, 24, 36, 37, 39, 40, 42, 43, 45, 46, 50, 51, 52, 53, 58, 59, 61, 62, 68, 69, 71, 72, 75, 76, 79, 80, 82, and 83. You can select all these by clicking **Select by group** and then selecting one of the boundaries in the list.

5 Click the **Load** tab.

6 Type `-p*nx_smsld` in the $\mathbf{F_x}$ (**Face load x dir**.) edit field.

7 Type `-p*ny_smsld` in the $\mathbf{F_y}$ (**Face load y dir**.) edit field.

8 Type `-p*nz_smsld` in the $\mathbf{F_z}$ (**Face load z dir**.) edit field.

9 Click **OK**.

MESH GENERATION

Click the **Initialize Mesh** button.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 Select **Time dependent** in the **Solver** list.

3 Type `0:0.05:1` in the **Times** edit field.

4 Select **UMFPACK** from the **Linear system solver** list.

5 Click **OK**.

6 Click the **Solve Manager** button.

7 Click the **Solve For** tab.

8 Select **Incompressible Navier-Stokes** from the **Solve for variables** list.

9 Click the **Solve** button.

10 Click the **Initial Value** tab.

11 Click the **Store Solution** button.

12 In the **Store Solution** dialog box, make sure that all time steps are selected, then click **OK**.

13 Select **Stored solution** in the **Values of variables not solved for and linearization point** area.

14 Select **All** from the **Solution at time** list.

15 On the **Solve For** page, select **Solid Stress-Strain** from the **Solve for variables** list.

16 Click **OK** to close the **Solver Manager** dialog box.

17 Click the **Solve** button on the Main toolbar.

POSTPROCESSING

Dynamics of a Bladder

Introduction

This model is related to a functional study of the main organs of the female pelvic floor and the phenomenon under investigation is the dynamic response of a bladder subjected to a particular load, in this case a short cough. From a clinical point of view and in particular in urology, a specific test is available in order to establish the degree of functionality of the bladder. This test and the related measurements are more properly defined as "urodynamics." This model provides an analysis of the displacements and stresses that the bladder can be subjected to during such a test.

The aim of the analysis is to reproduce the dynamics of the bladder, placed in the pelvic floor, when subjected to the movement caused by a short cough. The pelvic floor is characterized by the presence of numerous organs, bones, and muscles. To simplify the analysis, the model includes four principal components: the bladder itself (full of fluid), the muscles of the pelvic floor, and the two pelvis bones shown in Figure 5-7 below. One of the main difficulties of the model is the generation of the geometry of the various components from the CAT (computed axial tomography) scan images; here COMSOL Multiphysics' interface with COMSOL Script comes very handy.

Model Definition

Figure 5-7 shows the main components considered in the finite element model: the bladder, the pelvis muscles, and a portion of the bones of the pelvis to which the muscles are attached.

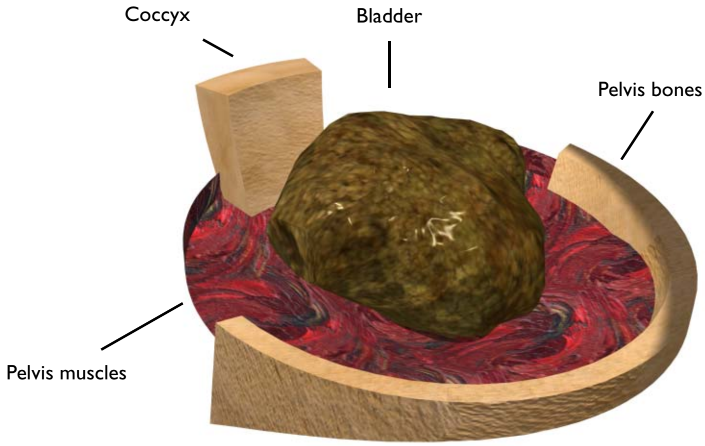


Figure 5-7: The bladder placed on the muscles of the pelvic floor, between the pelvic bones.

THE GEOMETRY

The geometry of the bladder is derived entirely from CAT scan images, illuminated on a diaphanoscope, in order to get the profiles of the bladder in various sections. The distance between the cross sections given in each of the scan is 1 cm.

The bladder can be considered as a little bag with variable thickness that at rest is 5 mm on average. For this particular case, a CAT scan was executed on a patient placed in a vertical position, with the bladder filled with fluid so that the thickness is 2 mm. Figure 5-8 shows a section of the bladder including the junction with the urethra. The thin three-dimensional structure is built up using contours of the cross sections of the internal and external walls of the bladder. Figure 5-9 shows an example of such a set of cross sections obtained from a CAT scan.

Figure 5-8: Cross section of the bladder including the junction with the urethra.

Each section is defined by 24 points taken with 15 degrees angles around the central axis of the structure. 22 sections (11 external and 11 internal) are collected in 22 spline curves and two groups of 11 splines are then joined to create two 3D geometries using the "loft" function. Two COMSOL Script files automatize this process.

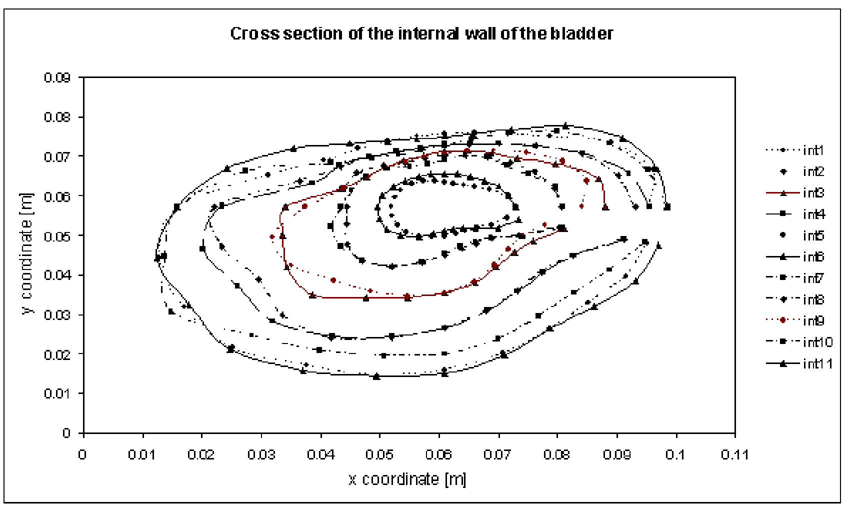


Figure 5-9: Contours of the internal walls of the bladder. The geometry is defined by 11 contour curves along the depth of the bladder.

The created geometry objects are exported to COMSOL Multiphysics from the COMSOL Script workspace. The remaining part of the model is created using the CAD tools in COMSOL Multiphysics. Figure 5-10 shows the resulting geometry as seen in the COMSOL Multiphysics user interface.

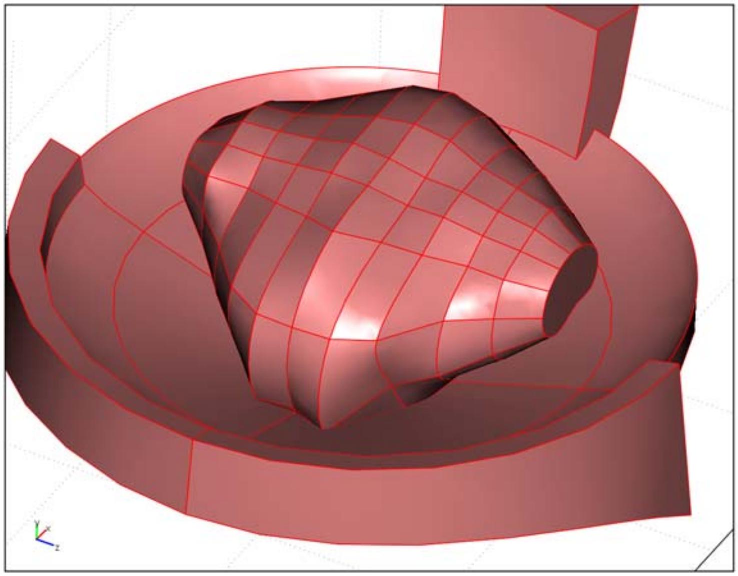


Figure 5-10: Bladder geometry used in the COMSOL Multiphysics model.

The resulting mesh consists of about 40,000 elements, which gives about 25,000 degrees of freedom. Figure 5-11 shows the mesh generated by COMSOL Multiphysics.

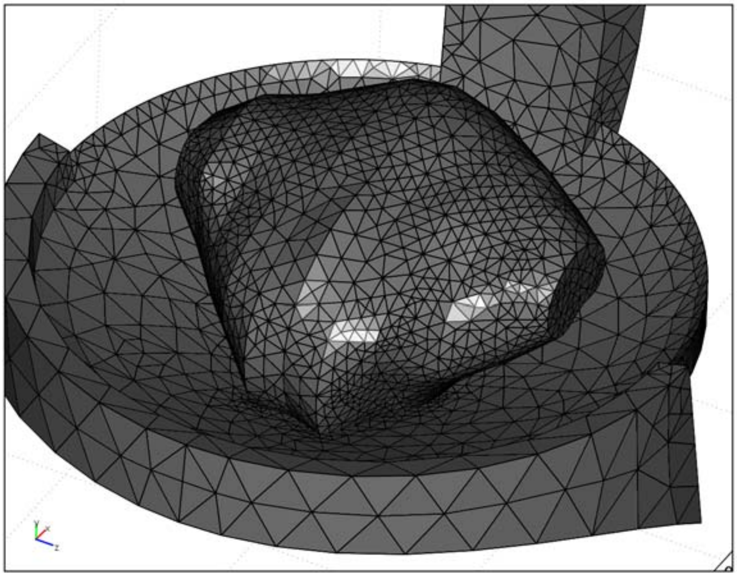


Figure 5-11: Finite element mesh created by COMSOL Multiphysics.

INPUT DATA

Biological tissues in general have peculiar constitutive properties and can also show time-dependent behavior. It is difficult to find literature data for such materials, but in the last few years a lot of research has been dedicated to the mechanical properties of biological tissue. One possible way of treating these materials is through the use of hyperelastic models. However, this model (where the mechanical loads are comparably small) uses linear elastic behavior for all materials. Also the fluid that fills the bladder is treated as an elastic material with a very small value of Young's modulus. This is acceptable in this case because only the mass contribution of the fluid during the dynamic analysis is of interest. The material properties are defined as follows:

Bladder

- Density = $956\ \mathrm{kg/m^3}$
- Young's modulus = $3.5\cdot10^5\ \mathrm{N/m^2}$
- Poisson's ratio $\nu = 0.45$

Pelvis muscles

- Density = 1200 kg/m^3
- Young's modulus = $1.162 \cdot 10^6$ N/m^2
- Poisson's ratio $\nu = 0.4$

Pelvis bones

- Density = 2570 kg/m^3
- Young's modulus = $1.0 \cdot 10^9$ N/m^2
- Poisson's ratio $\nu = 0.3$

Fluid

- Density = 1000 kg/m^3
- Young's modulus = $1.0 \cdot 10^3$ N/m^2
- Poisson's ratio $\nu = 0.495$

DYNAMIC ANALYSIS

The aim is to simulate the effect of two short coughs and the relaxation of the modeled components after such an excitation. The coughs are applied by imposing a displacement to the pelvis bones because these are almost rigid. The time duration of each single cough is 0.5 seconds (2 Hz), and the model assumes that a sinusoidal amplitude describes the applied displacements. A vertical and a rotational motion is applied for 1 second. During the remaining part of the time interval the imposed displacements are set to zero.

The critical damping ratio for low-frequency damping is estimated to be 0.2. The Rayleigh damping parameters are calculated assuming this damping ratio at two frequencies (1 Hz and 3 Hz) around the frequency of the cough (2 Hz). You can do this at the COMSOL Script command prompt with the following commands:

```
b = [0.2;0.2];
A = [1/(2*1*2*pi) 2*pi*1/2; 1/(2*3*2*pi) 2*pi*3/2];
% A*damp = b
damp = A\b;
alphadM = damp(1)
betadK = damp(2)
```

Resulting in $\alpha_{dM} = 1.88$ and $\beta_{dK} = 0.016$.

For more details about damping, see the section "Damping" on page 121 in the *Structural Mechanics Module User's Guide*.

Results

Figure 5-12 shows the vertical displacement after 0.14 seconds of the analysis, corresponding to the positive peak value of the displacement during the first cough.

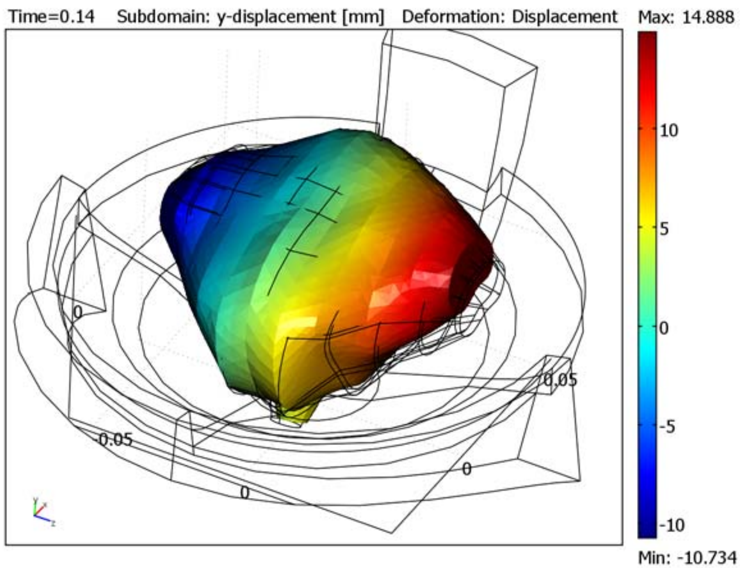


Figure 5-12: Y-displacement of the bladder taken as a snap shot after 0.14 s of the analysis.

Figure 5-13 shows a cross-sectional plot of the same displacement.

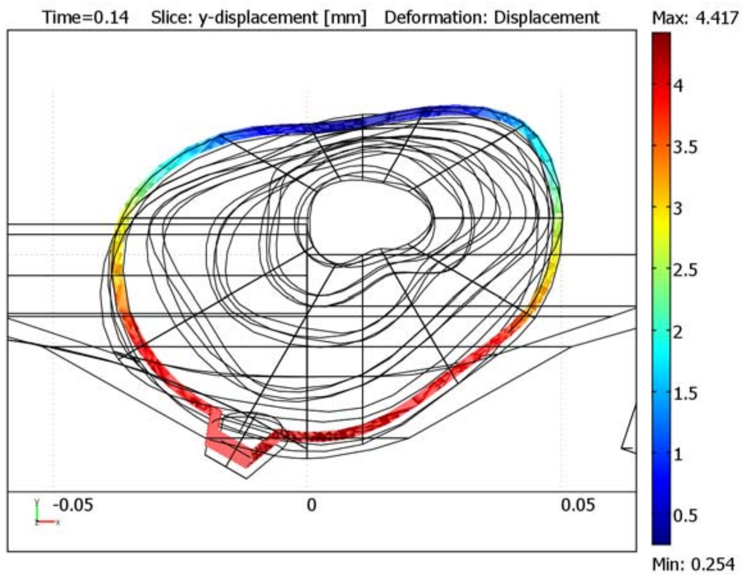


Figure 5-13: Vertical cross section of the displacement of the bladder after 0.14 s.

It is also of interest to determine the stresses in the bladder and their variation during the process. Figure 5-14 shows the distribution of the Von Mises stress after 0.14 seconds along the vertical cross section corresponding to the displacements in Figure 5-13.

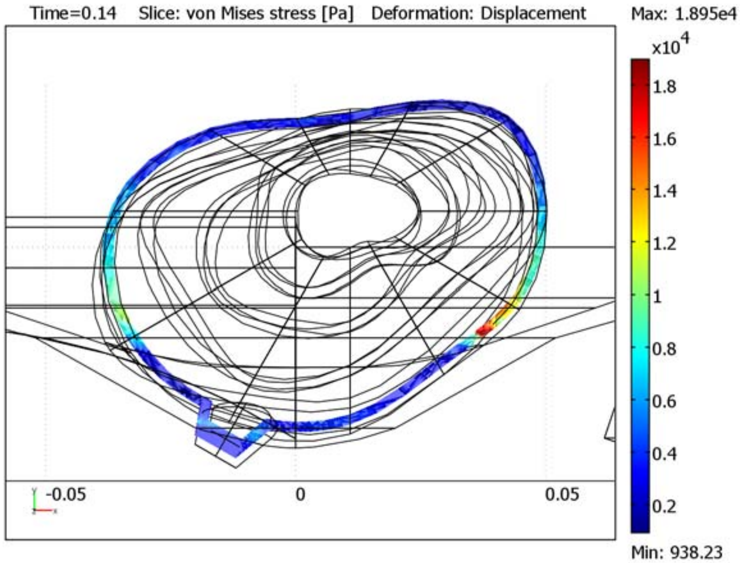


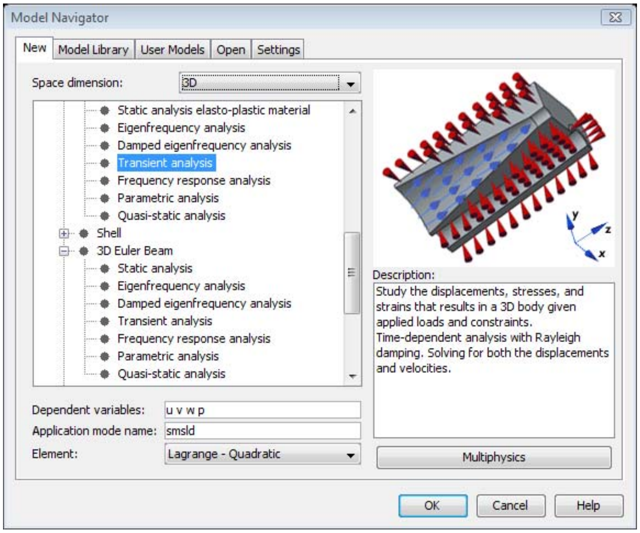
Figure 5-14: Von Mises stresses along a vertical cross section of the bladder after 0.14 s.

Model Library path: `Structural_Mechanics_Module/Bioengineering/bladder`

Modeling Using the Graphical User Interface

1 Click on the **COMSOL Multiphysics** icon to open the **Model Navigator**.
2 Select **3D** from the **Space dimension** list.
3 In the list of application modes, select the **Structural Mechanics Module>Solid, Stress-Strain>Transient analysis** application mode.

4 Click **OK**.



This presentation solves the problem using SI units (meters, seconds, Newtons, etc.).

OPTIONS AND SETTINGS

In the **Constants** dialog box, which you open from the **Options** menu, define constants according to the following table (the descriptions are optional); when finished, click **OK**.

NAME	EXPRESSION	DESCRIPTION
E_bone	1.0e9[Pa]	Young's modulus of bone
v_bone	0.3	Poisson's ratio of bone
rho_bone	2570[kg/m^3]	Density of bone
E_bladder	3.5e5[Pa]	Young's modulus of bladder
v_bladder	0.45	Poisson's ratio of bladder
rho_bladder	956[kg/m^3]	Density of bladder
E_fluid	1.0e3[Pa]	Young's modulus of fluid
v_fluid	0.495	Poisson's ratio of fluid
rho_fluid	1000[kg/m^3]	Density of fluid
E_muscle	1.162e6[Pa]	Young's modulus of muscle
v_muscle	0.4	Poisson's ratio of muscle

NAME	EXPRESSION	DESCRIPTION
rho_muscle	1200[kg/m^3]	Density of muscle
gravity	9.81[m/s^2]	Acceleration due to gravity

GEOMETRY MODELING

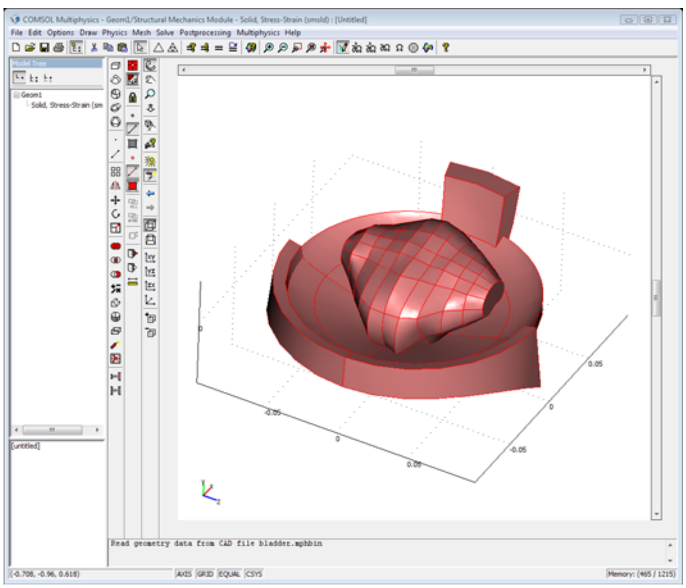
1 Select **Import** and **Cad Data From File** from the **File** menu.

2 Select **All files** in the **Files of type** list.

3 Select the file `bladder.mphbin` located in the folder `Structural_Mechanics_Module/Bioengineering` in the COMSOL installation directory and click **Import**.

4 Click the **Zoom Extents** button.

5 Click the **Headlight** button and rotate the geometry to get a proper view according to the figure below.



PHYSICS SETTINGS

Subdomain Settings

1 From the **Physics** menu, choose **Subdomain Settings**.

2 Select Subdomain 1 and set **Young's modulus** to `E_bone`, **Poisson's ratio** to `v_bone`, and **Density** to `rho_bone`.

3 Click the **Constraint** tab and select **Prescribed displacement** from the **Constraint condition** list.

4 Select the $\mathbf{R_x}$ and $\mathbf{R_y}$ check boxes to constrain the displacement.

5 Type `(t<=1)*((0.2*z+5[mm])*sin(2*pi*2[Hz]*t))` in the $\mathbf{R_y}$ edit field.

6 Click the **Load** tab and set $\mathbf{F_y}$ to `-rho_bone*gravity`.

7 Click the **Element** tab and select **Lagrange-Linear** elements.

8 Select Subdomain 2 and set **Young's modulus** to `E_muscle`, **Poisson's ratio** to `v_muscle`, and **Density** to `rho_muscle`.

9 Click the **Load** tab and set $\mathbf{F_y}$ to `-rho_muscle*gravity`.

10 Click the **Element** tab and select **Lagrange-Linear** elements.

11 Select Subdomain 3 and set **Young's modulus** to `E_bladder`, **Poisson's ratio** to `v_bladder`, and **Density** to `rho_bladder`.

12 Click the **Load** tab and set $\mathbf{F_y}$ to `-rho_bladder*gravity`.

13 Click the **Element** tab and select **Lagrange-Linear** elements.

14 Select Subdomain 4 and set **Young's modulus** to `E_fluid`, **Poisson's ratio** to `v_fluid`, and **Density** to `rho_fluid`.

15 Click the **Load** tab and set $\mathbf{F_y}$ to `-rho_fluid*gravity`.

16 Click the **Element** tab and select **Lagrange-Linear** elements.

17 Select Subdomain 5 and set **Young's modulus** to `E_bone`, **Poisson's ratio** to `v_bone`, and **Density** to `rho_bone`.

18 Click the **Constraint** tab and select **Prescribed displacement** from the **Constraint condition** list.

19 Select the $\mathbf{R_x}$, $\mathbf{R_y}$, and $\mathbf{R_z}$ check boxes to specify the displacement.

20 Type `(t<=1)*(5[mm]*sin(2*pi*2[Hz]*t))` in the $\mathbf{R_y}$ edit field.

21 Click the **Load** tab and set $\mathbf{F_y}$ to `-rho_bone*gravity`.

22 Click the **Element** tab and select **Lagrange-Linear** elements.

23 Click the **Damping** tab and select all subdomains to specify the same Rayleigh damping parameters to all subdomains.

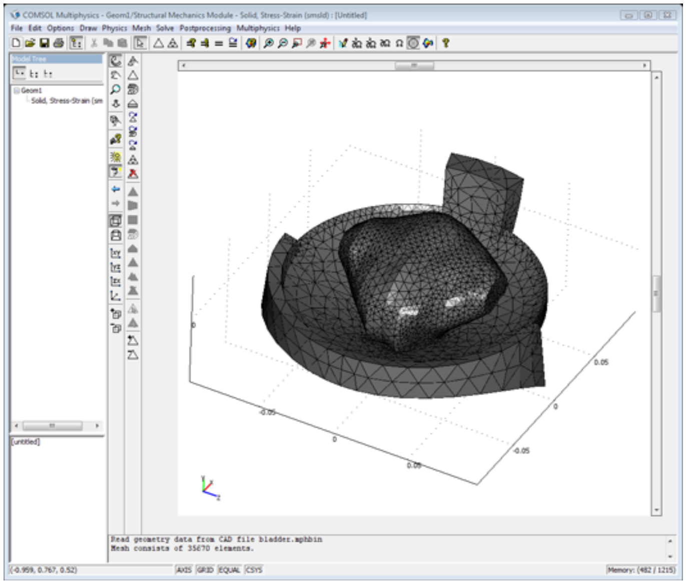
24 Set $\alpha_{\mathbf{dM}}$ to `1.88` and set $\beta_{\mathbf{dK}}$ to `0.016`.

25 Click **OK**.

26 Select **Properties** in the **Physics** menu and select **On** in the **Large deformation** list.

MESH GENERATION

1 Click the **Initialize Mesh** button.



COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 Select **Time dependent** analysis from the **Solver** list

3 Type `0:0.02:2` in the **Times** edit field.

4 Type `1e-3` in the **Relative tolerance** edit field.

5 Type `u 2e-7 v 3e-6 w 5e-7` in the **Absolute tolerance** edit field to specify individual tolerances for each displacement component.

6 Click **OK**.

7 Click the **Solve** button on the Main toolbar.

Biomedical Stent

Introduction

Percutaneous transluminal angioplasty with stenting is a widely spread method for the treatment of atherosclerosis. During this procedure a stent, which is a small tube like structure, is deployed in the blood vessel by using a balloon as an expander. Once the balloon-stent package is in place in the artery, the balloon is inflated to expand the stent. The balloon is then removed, but the expanded stent remains to act as a scaffold that keeps the blood vessel open.

Stent design is of significance for the procedure, since damage can be inflicted on the artery during the expansion process. One way this may happen is by the non-uniform deformation of the stent, where the ends expand more than the mid parts, which is also called dogboning. Foreshortening of the stent can also be damaging to the artery, and it can make the positioning difficult.

The dogboning is defined according to

$$\text{dogboning} = \frac{r_{\text{end}} - r_{\text{mid}}}{r_{\text{mid}}}$$

where r_{end} and r_{mid} are the radii at the end and middle of the stent, respectively.

The foreshortening is defined as

$$\text{foreshortening} = \frac{L_{\text{orig}} - L_{\text{load}}}{L_{\text{orig}}}$$

where L_{orig} is the original length of the stent and L_{load} is the deformed length of the stent.

To check the viability of a stent design, you can study the deformation process under the influence of a radial pressure which expands the stent. With a model you can easily monitor both the dogboning and foreshortening and draw conclusions on how to change the geometry design parameters for optimum performance.

Model Definition

Due to the circumferential and longitudinal symmetry of the stent, it is possible to model only one twenty-fourth of the geometry. But, for easier visualization of the deformation you can use one quarter of the geometry in the model.

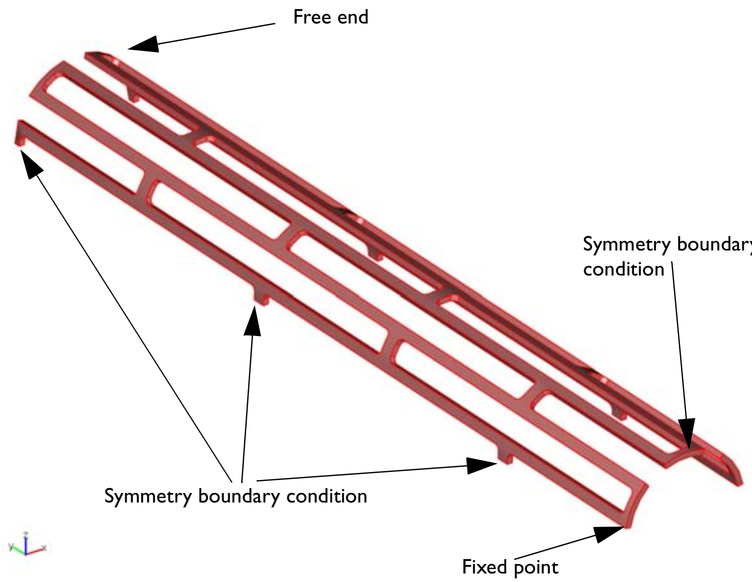


Figure 5-15: One quarter of the stent geometry.

MATERIAL

Assume that the stent is made of stainless steel with material parameters according to the following table.

MATERIAL PROPERTY	VALUE
Young's modulus	193 GPa
Poisson's ratio	0.3
Yield strength	300 MPa
Isotropic hardening modulus	2GPa

CONSTRAINTS

To prevent rigid body translation and rotation, apply the following constraints, which are also shown in Figure 5-15.

- Symmetry boundary conditions (normal displacements zero) for the boundaries lying in xy-plane and for the boundaries in the plane parallel to zx-plane going through the middle of the stent.
- Fix the rigid body translation in the remaining direction by constraining a point in the x direction.

LOADS

Apply a radially outward pressure on the inner surface of the stent. During loading, increase the pressure with the parametric solver to a maximum value of $p_{\mathrm{max}} = 0.3$ MPa. Follow by decreasing the load to zero to obtain the final shape of the deformed stent.

THE FINITE ELEMENT MESH

Use the predefined fine mesh size to mesh the geometry with the free mesher. The mesh created this way consists of approximately 7300 tetrahedral elements.

Results

The stent is expanded from an original diameter of 1.2 mm to a diameter of 3.6 mm in the mid section after unloading. The dogboning is 61% and the foreshortening is

-18%. These values are quite large and may require a change of the geometry parameters to improve the stent design.

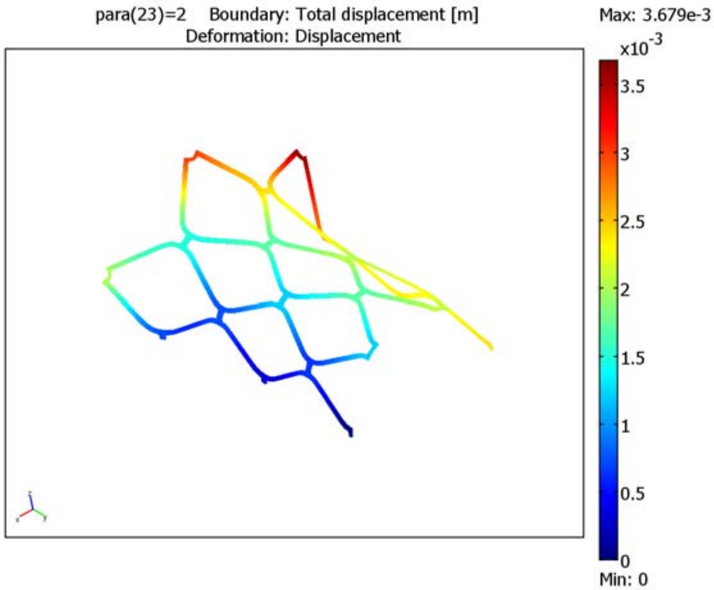


Figure 5-16: Deformed shape plot of the total displacement of the stent after unloading.

In the figure below you can see the diameter of the stent versus the load parameter. Parameter values above one signify the unloading phase. You can follow the deformation, which is initially linear and gets larger as the material is loaded beyond the yield stress. The unloading is purely elastic for this elasto-plastic material model with isotropic hardening.

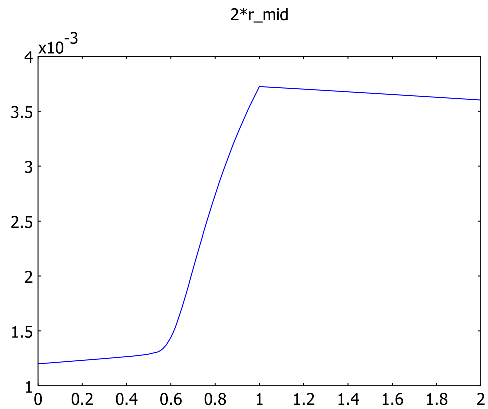


Figure 5-17: Diameter increase in the middle of the stent versus the load parameter.

You can also study the dogboning as the stent is expanded. In the following figure you can notice that the free end of the stent reaches the yield point sooner than the mid parts. As a result of this the dogboning parameter increases until the sections further in also start to deform plastically.

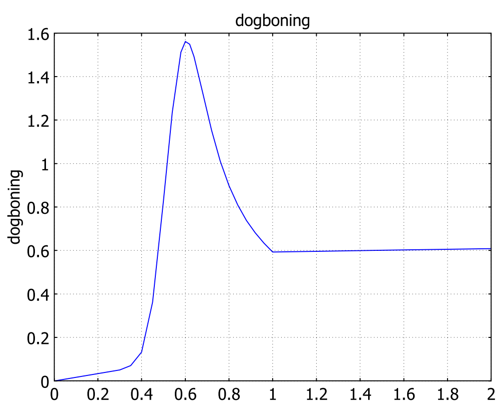


Figure 5-18: Dogboning versus the load parameter.

Model Library path: `Structural_Mechanics_Module/Bioengineering/biomedical_stent`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Click the **New** button or start COMSOL Multiphysics to open the **Model Navigator**.
2 Select **3D** from the **Space dimension** list.
3 Select **Structural Mechanics Module>Solid, Stress-Strain>Static analysis elasto-plastic material**.
4 Click **OK**.

IMPORT OF CAD GEOMETRY

1 On the **File** menu select **Import>CAD Data From File**.
2 In the **Files of type** list select **COMSOL Multiphysics file (*.mphtxt; *.mphbin; ...)**
3 Browse to the directory `models/Structural_Mechanics_Module/Bioengineering` located in the COMSOL installation directory and select the file `biomedical_stent.mphbin`.
4 Click **Import**.

OPTIONS AND SETTINGS

Constants

1 From the **Options** menu select **Constants**.
2 Enter the constants from the following table.

NAME	EXPRESSION	DESCRIPTION
Load_max	3e5	Maximum applied load to the stent
E	1.93e11	Young's modulus
nu	0.3	Poisson's ratio
S_yield	3e8	Yield stress
E_tan	2e9	Isotropic tangent modulus

3 Click **OK**.

Extrusion Coupling Variables

You can set up extrusion coupling variables to gain access to coordinates and displacements of points on the entire geometry.

1 Select **Options>Extrusion Coupling Variables>Point Variables**.
2 Use the following table to enter variables for selected points on the **Source** page. Click the **General transformation** option button for each variable, after entering both the name and expression.

POINTS	NAME	EXPRESSION
6	`x_mid`	`x`
6	`y_mid`	`y`
6	`u_mid`	`u`
272	`y_end1`	`y`
272	`z_end1`	`z`
272	`u_end1`	`u`
272	`v_end1`	`v`
272	`w_end1`	`w`
12	`x_end2`	`x`
12	`u_end2`	`u`
12	`w_end2`	`w`

3 Click the **Destination** tab.
4 Select first **x_mid** from the **Variable** list, then from the **Level** list select **Subdomain** and select the check box next to Subdomain 1 in the **Subdomain selection** list.
5 Repeat the previous step for all the variables you have entered under step 2.
6 Click **OK**.

Global Expressions

By defining the quantities you are interested in as global expressions you can access them for postprocessing or during the solution process.

1 From the **Options** menu select **Expressions>Global Expressions**.
2 Enter expressions according to the following table.

NAME	EXPRESSION
`foreshortening`	`(l_orig-l_load)/l_orig`
`l_orig`	`abs(y_mid-y_end1)`

NAME	EXPRESSION
l_load	l_orig+v_end1
dogboning	(r_end-r_mid)/r_mid
r_mid	r_orig+abs(u_mid)/2
r_orig	abs(x_mid)
r_end	sqrt((z_end1+w_end1)^2+(dist12/2)^2)
dist12	sqrt((w_end2-w_end1)^2+(u_end2-u_end1+2*x_end2)^2)

3 Click **OK**.

PHYSICS SETTINGS

Application Mode Properties

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

2 In the **Large deformation** list box select **On**.

3 Click **OK**.

Subdomain Settings

1 Select **Physics>Subdomain Settings** to open the **Subdomain Settings** dialog box.

2 Select Subdomain 1.

3 On the **Material** page, select **Elasto-plastic** from the **Material model** list.

4 In the **E** edit field enter `E`.

5 In the ν edit field enter `nu`.

6 Click the **Elasto-plastic material data** button.

7 In the σ_{ys} edit field enter `S_yield`.

8 In the E_{Tiso} edit field enter `E_tan`.

9 Click **OK**.

10 Click **OK**.

Boundary Conditions

By applying the symmetry boundary conditions you can prevent rigid body translation in the *y* and *z* direction and rotation around all coordinate axes.

1 Select **Physics>Boundary Settings**.

2 From the **Boundary selection** list select Boundaries 2, 5, 8, 10, 42, 86, 130, 144, 145, and 146.

3 On the **Constraint** page select **Symmetry plane** from the **Constraint condition** list.

4 Select Boundary 28.

5 On the **Load** page, from the **Coordinate system** list box select **Tangent and normal coord. sys. (t_1, t_2, n)**.

6 In the **F_n** edit field enter `-Load_max*((para<=1)*para+(para>1)*(2-para))`.

7 Click **OK**.

Point Settings

To fix the rigid body translation in the remaining x direction constrain a point in this direction.

8 Select **Physics>Point Settings**.

9 From the **Point selection** list select Point 288.

10 On the **Constraint** page make sure that the **Standard notation** button is selected.

11 Select the **R_x** check box and check that the value in the corresponding edit field is zero.

12 Click **OK**.

MESH GENERATION

1 From the **Mesh** menu, choose **Free Mesh Parameters**.

2 Select **Fine** from the **Predefined mesh sizes** list.

3 Click the **Remesh** button, then click **OK**.

COMPUTING THE SOLUTION

Solver Parameters

1 Click the **Solver Parameters** button on the Main toolbar.

2 On the **General** page, in the **Parameter name** edit field enter `para`.

3 In the **Parameter values** edit field enter `0 0.3 0.35 0.4 0.45 0.5 0.54 0.58 0.6 0.62 0.64:0.04:1 1.05 1.5 2`.

4 Click **OK**.

Probe Plot

To monitor the deformation of the stent during the solution process, you can create a probe plot. You can see this diagram, updated for each parameter step, on the **Plot** page of the **Progress** dialog box, that COMSOL Multiphysics displays while the solver is running.

1 Select **Postprocessing>Probe Plot Parameters**.

2 In the **Probe Plot Parameters** dialog box click the **New** button.

3 In the dialog box that opens select **Global** from the **Plot type** list box.

4 In the **Plot name** edit field enter `Stent diameter plot`.

5 Click **OK**.

6 In the **Expression** edit field type `2*r_mid`.

7 Click **OK**.

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

POSTPROCESSING AND VISUALIZATION

When the solution is done the plot of the stent diameter versus the solver parameter is automatically displayed in a separate figure window. You can also visualize the deformation of the stent after unloading.

1 Switch back to the COMSOL Multiphysics window.

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

3 On the **General** page deselect the **Slice** check box under the **Plot type** area.

4 Deselect the **Geometry edges** check box.

5 Select the **Boundary** check box.

6 Select the **Deformed shape** check box.

7 Click **OK**.

You can also display the foreshortening and dogboning of the stent.

8 Select **Postprocessing>Data Display>Global**.

9 In the **Expression** edit field type `dogboning`.

10 Click **Apply** to display the value in the message log.

11 In the **Expression** edit field type `foreshortening`.

12 Click **OK** to display the value in the message log and close the **Global Data Display** dialog box.

6

Civil Engineering Models

This chapter contains an analysis of a truss bridge.

Pratt Truss Bridge

The following procedure steps through the analysis of a bridge using the Structural Mechanics Module.

Model Definition

The geometry of the bridge is inspired by a common bridge type called a Pratt truss bridge. You can identify a Pratt truss by its diagonal members, which (except for the very end ones) all slant down and in toward the center of the span. All the diagonal members are subject to tension forces only, while the shorter vertical members handle the compressive forces. This allows for thinner diagonal members resulting in a more economic design.

A *truss structure* supports only tension and compression forces in its members and you would normally model it using bars, but as this model uses 3D beams it also includes bending moments to some extent in a *frame structure*. In the model, shell elements represent the roadway.

ANALYSIS TYPES

The model includes three different analyses of the bridge:

- The goal of the first analysis is to evaluate the stress and deflection fields of the bridge when exposed to a pure gravity load.
- The goal for second analysis is to evaluate stress and deflection fields with a load corresponding to a truck parked at the middle of the bridge.
- Finally, an eigenfrequency analysis shows the eigenfrequencies and eigenmodes of the bridge.

LOADS AND CONSTRAINTS

To prevent rigid body motion of the bridge, it is important to constrain it properly. All translational degrees of freedom are constrained at the left-most horizontal edge. Constraints at the right-most horizontal edge prevent if from moving in the y and z directions but allow the bridge to expand or contract in the x direction. Figure 6-1 shows the bridge geometry with constraint symbols and load symbols for the gravity load on the roadway.

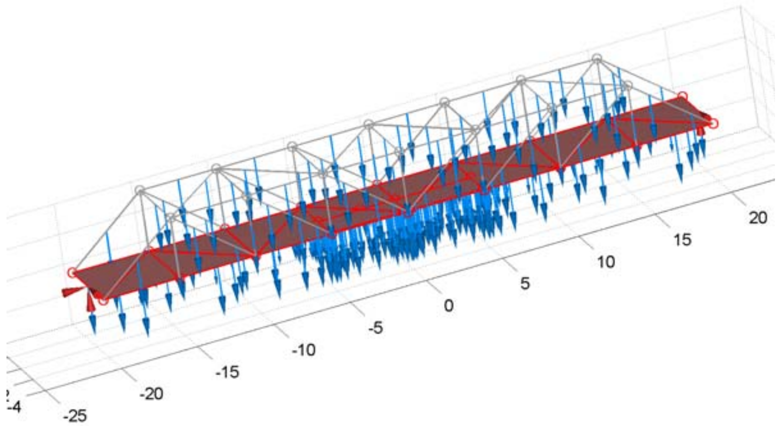


Figure 6-1: The bridge geometry together with constraints symbols (red arrows) and gravity load/truck load symbols on the roadway and frame (blue arrows).

MATERIAL PROPERTIES

The material in the frame structure is steel and the cross-section data for the frame beam members is for a standard HEA 100 beam (H-beam). The roadway material is concrete.

The frame members are orientated so that their local *xy*-planes are parallel to the global *xy*-plane (see Figure 6-2).

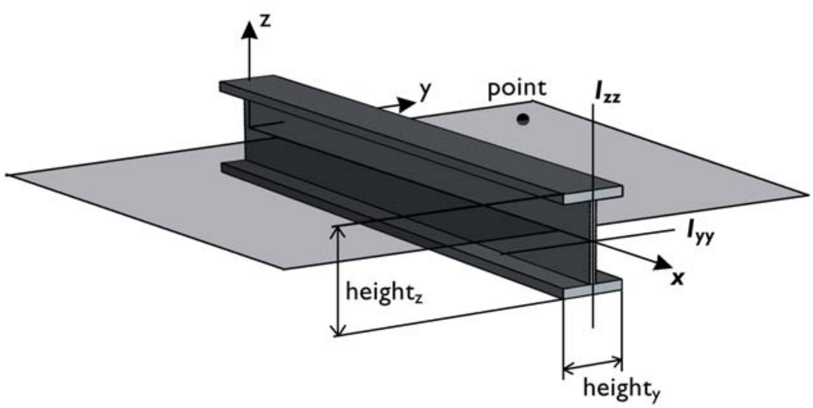


Figure 6-2: The local beam coordinate system.

Results

Figure 6-3 and Figure 6-4 illustrate the result. The distribution of axial stresses demonstrates the functionality of the frame: The interplay of members in tension and compression achieve the load bearing function. The upper horizontal members are in compression and the lower in tension. All the diagonal members are subject to tension forces only while the shorter vertical members handle the compressive forces. Figure 6-3 also shows that the maximum deflection amounts to 9.8 cm.

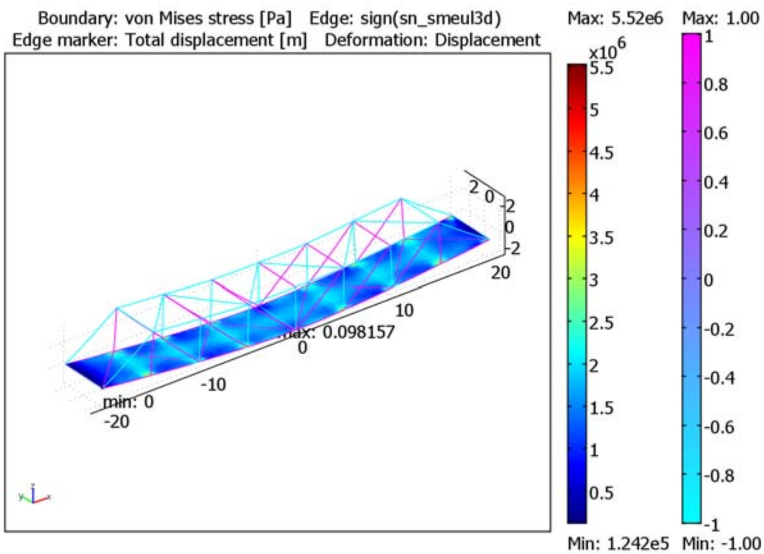


Figure 6-3: Axial stresses in the frame and von Mises stresses in the roadway (magenta members in tension, blue members in compression).

To evaluate the added deflection that a truck causes, the second analysis includes a parked truck in the middle of the bridge on one side of the road. The truck has a mass of 10,000 kg and measures 10 meters in length and 2 meters in width. The maximum deflection of the truss bridge is now 11.8 cm compared to the earlier unloaded value of 9.8 cm (see Figure 6-4).

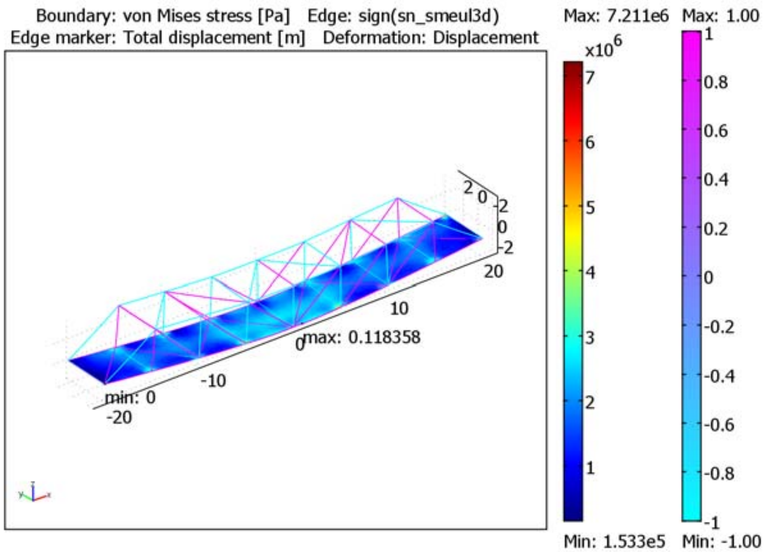


Figure 6-4: Truck load analysis: Axial stresses in the frame (magenta members in tension, blue members in compression).

The study of eigenfrequencies is important with respect to the excitation and frequency content from various loads such as wind loads and earthquakes.

Figure 6-5 shows the fourth eigenmode of the bridge.

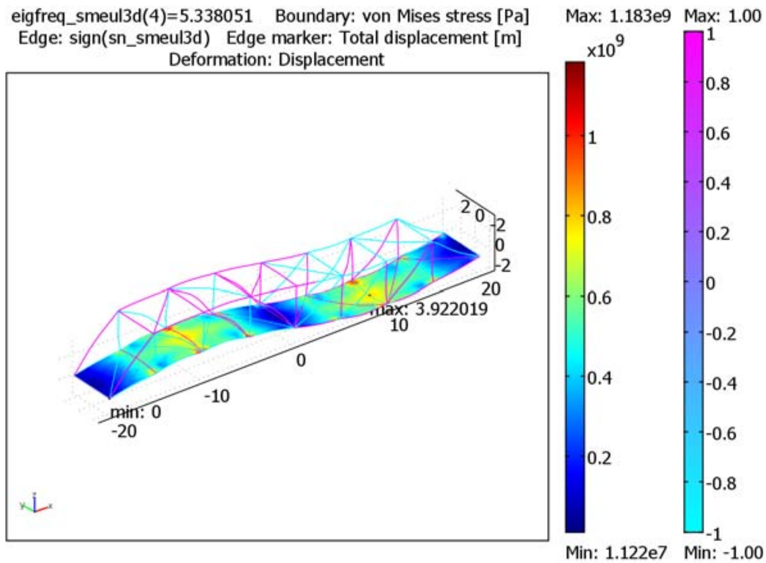


Figure 6-5: The fourth eigenmode.

Model Library path: `Structural_Mechanics_Module/Beam_Models/bridge`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

This model uses Euler beam elements to model the frame members.

1 Select **3D** in the **Space dimension** list.

2 In the list of application modes, select **Structural Mechanics Module>3D Euler Beam> Static analysis**.

3 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Constants** to specify the gravitational constant `gc` and the mass `m_Truck` of the largest truck that is allowed on the truss bridge:

NAME	EXPRESSION	DESCRIPTION
m_Truck	1e4[kg]	Mass of truck
gc	9.81[m/s^2]	Acceleration due to gravity

2 Click **OK**.

GEOMETRY MODELING

Start the modeling by adding a suitable work plane to create the profile of the bridge. Use a work plane at $y = -2.5$.

1 Select **Work-Plane Settings** from the **Draw** menu. Create a workplane from the **Quick** page by selecting the **z-x** option button and typing `-2.5` in the **y** edit field.

2 Change the grid spacing to `1` in the **Axes/Grid Settings** dialog box that you open from the **Options** menu.

3 Start by double-clicking **SOLID** in the Status bar to create curve objects by default when drawing. On the **Draw** menu, select **Draw Objects>Line** and make a closed polygon with the corners in (–2, 20), (–2, –20), (2, –15), and (2, 15). Finalize the polygon by right-clicking.

4 Select **Extrude** from the **Draw** menu, and change the extrusion distance to `5`. This will create the extruded profile of the bridge model.

5 Now add a work plane on one of the sides of the bridge. Do this by first clicking **Add** in the **Work-Plane Settings** dialog box, and then selecting Vertices 1, 7, and 3 on the extruded object in that order. A vertex is selected by clicking **>>** on the **Vertices** page in the **Work-Plane Settings** dialog box. The order of the vertices defines the orientation of the plane.

6 Change the grid spacing to 1 in the new work plane, and change to curve object default by double-clicking **SOLID**.

7 Draw an open line object with corners in (5, 0), (5, 4), (10, 0), (10, 4), (15, 0), (15, 4), (20, 0), and (20, 4).

8 Select mirror and enter `20` in the edit field for the x coordinate of **Point on line**. This should give the other half of the frame of the bridge.

9 Select both objects and click the **Coerce to Curve** button on the Draw toolbar.

10 Select **Embed** from the **Draw** menu. To embed the curve object in 3D, accept the default choices in the dialog box by clicking **OK**.

11 Create a copy of the 3D curve object, and enter the *y*-displacement 5 when pasting.

12 Next create a work plane on the bottom of the bridge. Open **Work-Plane Settings** dialog box, click **Add**, and then define the new work plane in the **Edge Angle** page by selecting Edge 1 and Face 2 of the extruded object.

13 Double-click **SOLID** in the Status bar, click the **Line** toolbar button, and draw an open polygon curve with corners in (0, 40), (5, 35), (0, 30), (5, 25), (0, 20), (5, 15), (0, 10), (5, 5), and (0, 0).

14 Draw a line from (0, 35) to (5, 35). Click the **Array** button and enter 7 as array size in the *y* direction with a displacement of -5.

15 Select all curves (including lines), then click **Coerce to Curve**.

16 From the **Draw** menu choose **Embed**, then click **OK** in the dialog box that opens.

17 Once again, click **Add** in the **Work-Plane Settings** dialog box, then define the plane by selecting Edge 6 and Face 3 on the **Edge Angle** tab.

18 Double-click **SOLID** in the Status bar.

19 Create the open polygon with corners in (0, 30), (5, 25), (0, 20), (5, 15), (0, 10), (5, 5), and (0, 0).

20 Draw a line from (0, 25) to (5, 25). Then click the **Array** button and enter 5 as array size in the *y* direction with a displacement of -5.

21 Select all curves, then click **Coerce to Curve**.

22 Embed the object by choosing **Draw>Embed** and then clicking **OK**.

23 In 3D, select the extruded object and click **Split**.

24 Select all objects but the planar face forming the road of the bridge. Use Ctrl-click to do this. Click **Coerce to Curve** to generate one curve object out of this.

25 In Geom4, Shift-click the **Rectangle/Square** button on the Draw toolbar to create a rectangle with width 2, length 10, and corner in (0.25,15). Then select **Embed** from the **Draw** menu to embed the rectangle.

Bridge Under Gravity Load

MODEL NAVIGATOR

1 Open the **Model Navigator** from the **Physics** menu. Select **3D** in the **Space dimension** list.

2 In the list of application modes, select **Structural Mechanics Module>Shell>Static analysis**.

3 Click **Add**, then click **OK**.

PHYSICS SETTINGS

Edge Settings—Shell

The goal of the first analysis is to evaluate the stress and deflection fields of the bridge when it is exposed to gravity only. To prevent rigid-body motion of the bridge, it is important to constrain it properly.

1 From the **Physics** menu, choose **Edge Settings**.

2 Click the **Constraint** tab and specify the following constraints:

SETTINGS	EDGE 1	EDGE 106
Constraint condition	Pinned	Prescribed displacement
R_y		0
R_z		0

3 Click **OK**.

Boundary Settings—Shell

Gravity acts on the entire road.

1 From the **Physics** menu, choose **Boundary Settings**.

2 Select all shell domains by selecting one boundary, and then pressing Ctrl+A.

3 Click the **Material** tab.

4 Click the **Load** button.

5 Select **Concrete** in the **Materials** list in the **Materials/Coefficients Library** dialog box.

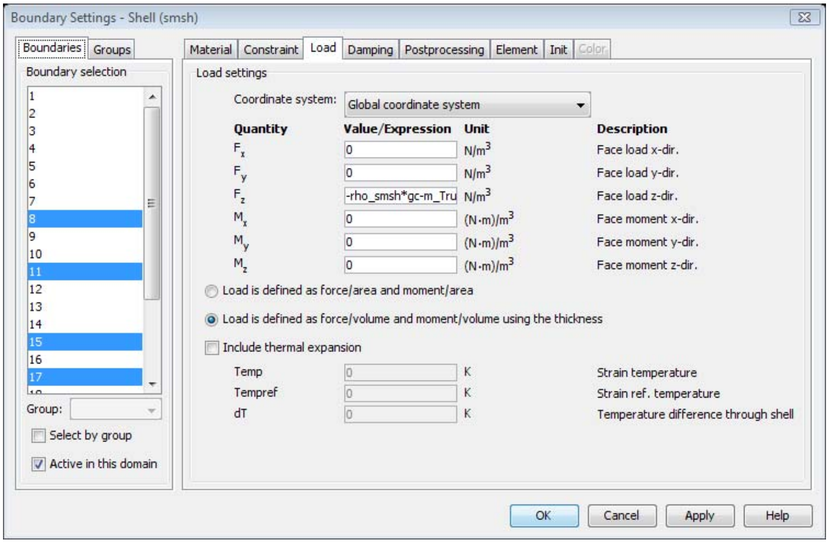
6 Click **OK**.

7 Type `0.2` in the **Thickness** edit field.

8 Click the **Load** tab.

9 In the $\mathbf{F_z}$ edit field type `-rho_smsh*gc`.

10 Click the **Load is defined as force/volume and moment/volume using the thickness** option button.



11 Click **OK**.

Edge Settings—3D Euler Beam

1 From the **Multiphysics** menu, choose **1 Geom1: 3D Euler Beam (smeul3d)**.

2 From the **Physics** menu, choose **Edge Settings**.

3 Select all edges by selecting one boundary, and then pressing Ctrl+A.

4 Click the **Material** tab.

5 Click the **Load** button.

6 Select **Structural steel** in the **Materials** list in the **Materials/Coefficients Library** dialog box.

7 Click **OK**.

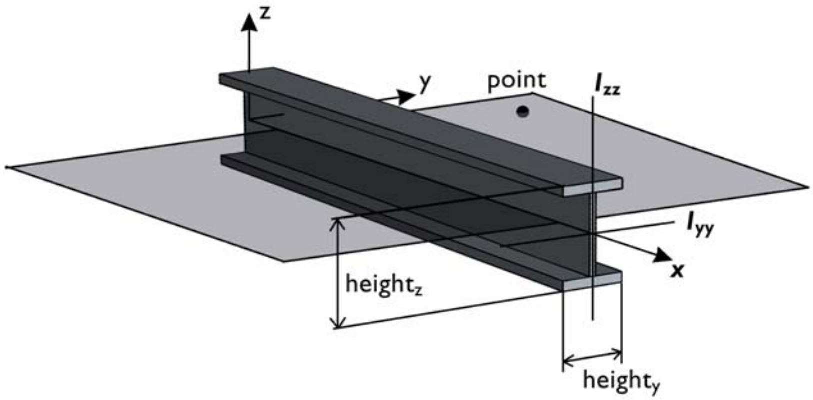
8 Click the **Cross Section** tab.

9 Click the **Load** button to open the **Cross-Section Library** dialog box.

10 In the **Cross sections** list, select the HEA 100 beam cross section.

11 Click **OK**.

12 Next, specify the frame members local x-y plane, that is, the orientation of the beams about their local x-axis (see the figure below) using a point. Enter the point x-, y-, and z-coordinates in the **localxp**, **localyp**, and **localzp** edit fields.



13 For each side, select all edges and assign local point coordinates according to the following tables:

EDGE	10, 12, 13, 18, 23, 25, 28, 30, 34, 36, 37, 46, 56, 57, 65, 66, 75, 76, 77, 84, 90, 91, 95, 96, 100	1,2,4, 5, 8, 9, 15, 16, 20, 21, 22, 27, 32, 33, 38, 40, 43, 44, 47, 49, 52, 53, 54, 58, 60, 63, 67, 70, 72, 73, 78, 79, 81, 82, 86, 87, 89, 93, 98, 99, 103, 104, 106
localxp	1	1
localyp	5	5
localzp	2	-2

EDGE	3, 7, 11, 19, 24, 31, 35, 51, 55, 71, 74, 85, 88, 97, 101	6, 14, 17, 26, 29, 42, 45, 62, 64, 80, 83,92, 94, 102, 105
localxp	1	1
localyp	-2.5	2.5
localzp	3	3

14 Click the **Load** tab.

15 Press Ctrl+A to select all edges and type `-rho_smeul3d*A_smeul3d*gc` in the $\mathbf{F_z}$ edit field.

When you added the smaller rectangle for the truck load that resulted in some additional edges. Deactivate these edges from the beam application mode:

16 Select Edges 39, 41, 48, 50, 59, 61, 68, and 69.

17 Deactivate these edges by clearing the **Active in this domain** check box.

18 Click **OK**.

MESH GENERATION

1 From the **Multiphysics** menu, choose **2 Geom1: Shell (smsh)**.

2 From the **Mesh** menu, choose **Free Mesh Parameters**.

3 Select **Extremely fine** from the **Predefined mesh sizes** list.

4 Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot illustrates the total displacement by superposition of edge and color plots. In addition, visualize the axial stresses in the individual members as follows:

1 From the **Postprocessing** menu, choose **Plot Parameters**.

2 Click the **General** tab and clear the **Geometry edges** check box. Select the **Deformed shape** check box.

3 Because of the beam elements, the automatic element refinement sets the refinement level to 6, which is unnecessarily high considering the fine mesh in the shell. To the right of **Element refinement**, clear the **Auto** check box and type 2 in the edit field.

4 Click the **Edge** tab. In the **Predefined quantities** list, select **3D Euler Beam (smeul3d)>Axial stress**.

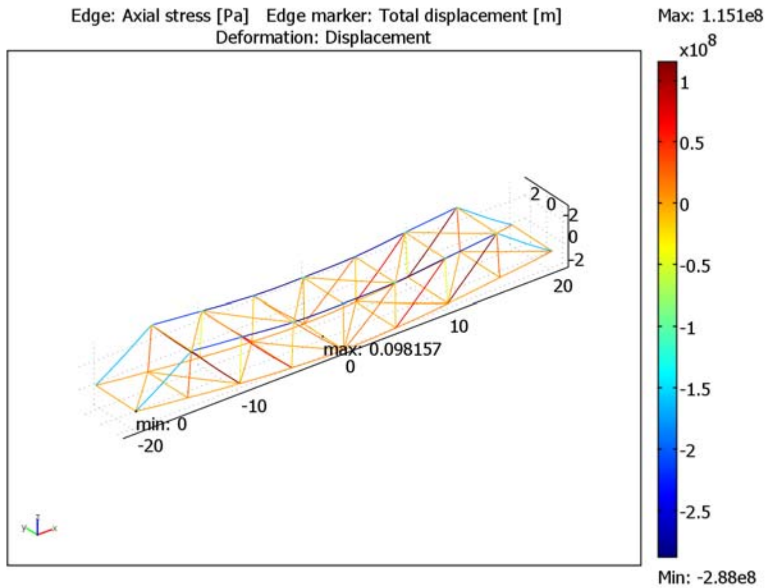
5 Click the **Max/Min** tab.

6 Select the **Max/min marker** check box.

7 On the **Edge Data** tab, select the **Edge max/min data** check box and select **3D Euler Beam (smeul3d)>Total displacement** in the **Predefined quantities** list.

8 Click **Apply**.

The following figure illustrates the result. You can see that the maximum deflection amounts to 9.8 cm. The distribution of axial stresses demonstrates the functionality of the frame: The load bearing function is achieved by the interplay of members in tension and compression. The upper horizontal members are in compression and the lower in tension.



Furthermore, it may be of interest to visualize the equivalent (von Mises) stress imposed on the road:

1 Open the **Plot Parameters** dialog box.

2 Click the **General** tab.

3 Select the **Boundary** check box to activate a plot on the boundary (face) representing the road.

4 Click the **Boundary** tab.

5 Select **Shell (smsh)>von Mises stress** in the **Predefined quantities** list.

6 Click **Apply**.

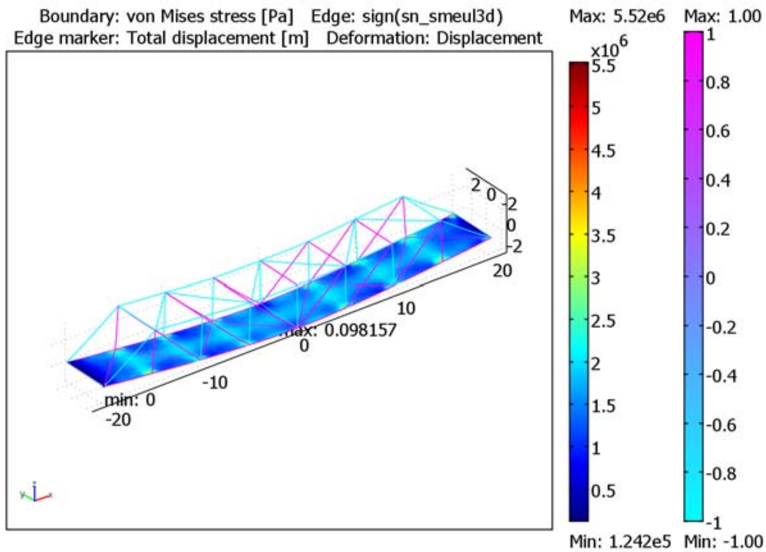
In addition, to visualize the distribution of members in tension and compression the stress range for the beam axial stress plot is changed to show the sign of the axial stress in the individual members (tension—positive stress values, compression—negative stress values).

1 Click the **Edge** tab.

2 Type `sign(sn_smeul3d)` in the **Expression** edit field.

3 In the **Colormap** list select **cool**.

4 Click **Apply**.



Truck on the Bridge

To evaluate the added deflection that a truck causes, park it in the middle of the bridge on one side of the road. The truck has a mass of 10,000 kg and measures 10 meters in length and 2 meters in width.

PHYSICS SETTINGS

Boundary Conditions

1 From the **Physics** menu, choose **Boundary Settings**.

2 Click the **Load** tab.

3 Select Boundaries 8, 11, 15, and 17 in the **Boundary selection** list (the area representing the truck) and type `-rho_smsh*gc-m_Truck*gc/(thickness_smsh*10[m]*2[m])` in the $\mathbf{F_z}$ edit field.

4 Click **OK**.

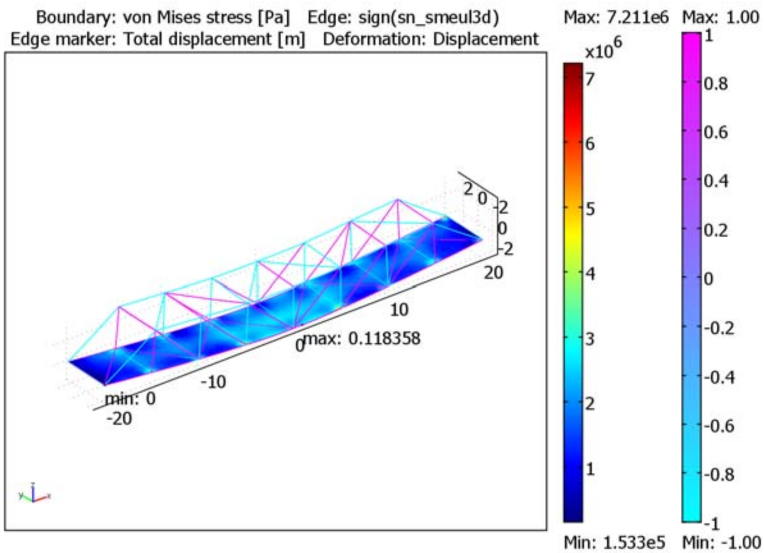
COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The plot shows the same properties as for the previous analysis.

The maximum deflection of the truss bridge is now 11.8 cm compared to the earlier unloaded value of 9.8 cm.



Eigenfrequencies of the Bridge

An eigenfrequency analysis is important with respect to impact of the target environment.

COMPUTING THE SOLUTION

1 From the **Solve** menu, choose **Solver Parameters**.

2 In the **Analysis** list, choose **Eigenfrequency**.

Change the analysis type for the Shell application mode to eigenfrequency as well.

3 From the **Physics** menu, choose **Properties**.

4 Select **Eigenfrequency** from the **Analysis type** list.

5 Click **OK** to close the **Application Mode Properties** dialog box.

6 Open the **Solver Parameters** dialog box from the **Solve** menu. On the **General** page, type 20 in the **Desired number of eigenfrequencies** edit field and -1 in the **Search for eigenfrequencies around** edit field.

7 Click **OK**.

8 Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

Review the eigenfrequencies and plot a mode shape:

1 From the **Postprocessing** menu, choose **Plot Parameters**.

2 Investigate the eigenmodes by selecting the corresponding eigenfrequencies in the **Eigenfrequency** list on the **General** page. Select the eigenfrequency at approximately 5.3 Hz and click **Apply**.

Figure 6-5 on page 250 shows the result of the plot.

7

Contact and Friction Models

This section contains examples of contact and friction models that you can study using the Plane Stress, Plane Strain, and Solid Stress-Strain application modes in the Structural Mechanics Module.

Sliding Wedge

Introduction

This is a benchmark model for contact and friction described in the NAFEMS publication in Ref. 1. An analytical solution exists, and this model includes a comparison of the COMSOL Multiphysics solution against the analytical solution.

Model Definition

A contactor wedge under the gravity load G is forced to slide due to a boundary load, F, over a target wedge surface, both infinitely thick (see Figure 7-1). Horizontal linear springs are also connected between the left vertical boundary of the contactor and the ground. The total spring stiffness is K.

This is a large sliding problem including contact pressure, for a constant contact area, and friction. A boundary contact pair is created and the contact functionality of the Structural Mechanics module is used to solve the contact/friction problem. Friction is modeled with the Coulomb friction model.

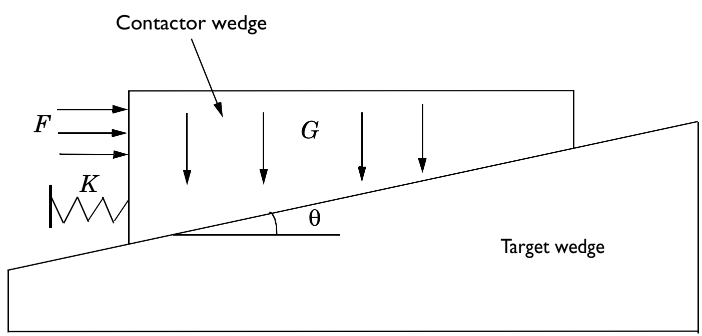


Figure 7-1: Sliding wedge with linear springs and a boundary load.

The aim of this benchmark is to calculate the horizontal sliding distance and compare it with an elementary statics calculation.

The overall spring stiffness, K, is determined to $K = 563.9$ N/m by an elementary statics calculation for the following values: the sliding distance $x = 1.0$ m, the horizontal force $F = 1500$ N, the total vertical gravity load $G = 3058$ N (which divided by the wedge volume gives a load/volume of $764.5\ \mathrm{N/m^3}$), the wedge angle $\tan\theta = 0.1$, and the coefficient of friction $\mu = 0.2$. The overall spring stiffness is determined to $K = 563.9\ \mathrm{N/m}$.

The spring is modeled as an x-displacement dependent boundary load, that is, a deformation-dependent load.

A suitable element type for this model is the linear quadrilateral element (see Figure 7-2).

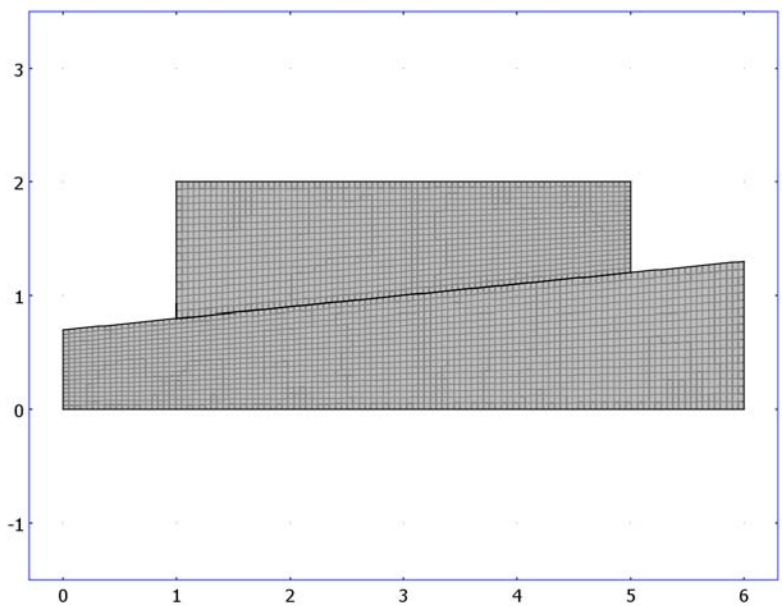


Figure 7-2: Linear quadrilateral elements is used to mesh the model.

The total number of elements in this model is 3686 and the number of degrees of freedom is 8554.

A suitable solver for contact/friction problems is the parametric solver where the gravity load G and the boundary load F are increased incrementally.

This model uses manual scaling of the solution components to reduce the solution time. You can find further explanation of manual scaling for contact models on page 129 of the *Structural Mechanics Module User's Guide*.

Results and Discussion

The model is in good agreement with Ref. 1. Figure 7-3 shows the result from the analysis.

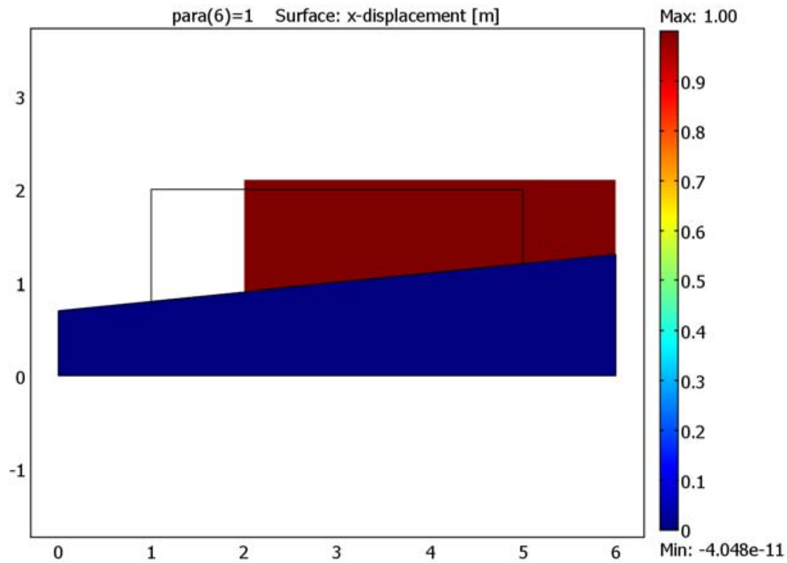


Figure 7-3: A *surface plot of the x-displacement of the contactor wedge.*

Reference

1. Feng Q., *NAFEMS Benchmark Tests for Finite Element Modelling of Contact, Gapping and Sliding*. NAFEMS Ref. R0081, UK, 2001.

Model Library path: `Structural_Mechanics_Module/Contact_and_Friction/sliding_wedge`

MODEL NAVIGATOR

1 On the **New** page select **2D** from the **Space dimension** list.

2 From the list of application modes select **Structural Mechanics Module>Plane Strain>Static analysis**, then click **OK**.

OPTIONS AND SETTINGS

From the **Options** menu select **Constants** and enter the following constant names, expressions, and descriptions (the descriptions are optional). When finished, click **OK**.

NAME	EXPRESSION	DESCRIPTION
G	3058[N]	Gravity load
V_wedge	4[m^3]	Volume of wedge
gravity	G/V_wedge	Gravity load/volume
mu	0.2	Coefficient of friction
F	1500[N]	Boundary load (force)
L_edge	1.2[m]	Length of left vertical edge
F_per_unit_area	F/L_edge	Force per unit area
K	563.9[N/m]	Overall spring stiffness
K_per_length	K/L_edge	Spring stiffness per length

GEOMETRY MODELING

1 Shift-click the **Line** button on the Draw toolbar to open the **Line** dialog box.

2 In the **x** edit field type `0 0 6 6` and in the **y** edit field type `0 0.7 1.3 0`.

3 From the **Style** list select **Closed polyline (solid)**, then click **OK**.

4 Shift-click the **Line** button on the Draw toolbar.

5 In the **x** edit field type `1 1 5 5` and in the **y** edit field type `0.8 2 2 1.2`.

6 From the **Style** list select **Closed polyline (solid)**, then click **OK**.

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

8 Select **Use Assembly** from the **Draw** menu.

9 Check your geometry against the figure below.

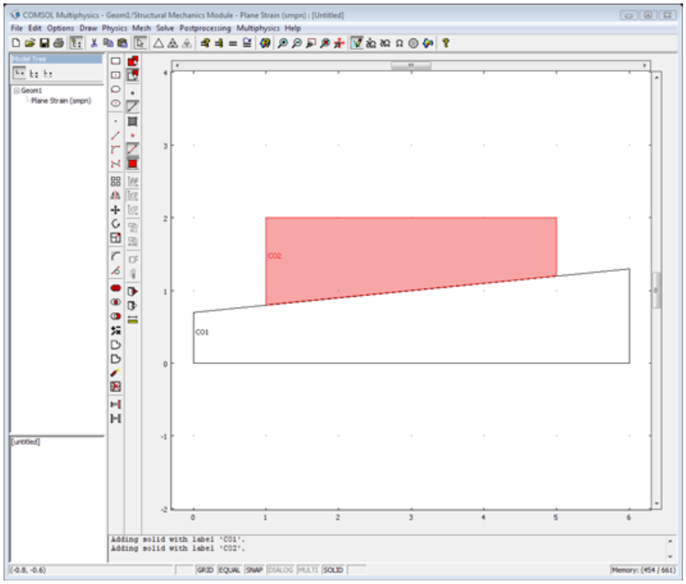


Figure 7-4: Sliding wedge geometry.

PHYSICS SETTINGS

Subdomain Settings

Click **Physics>Subdomain Settings** and specify the material properties and body load according to the following table:

SUBDOMAIN	PAGE	SETTINGS/INPUT	
1, 2	Material	E	`206e9`
		ν	`0.3`
2	Load	F_y	`-gravity*para`
1,2	Element	Predefined element	`Lagrange - Linear`

Boundary Conditions

Contact Pairs

1 Select **Physics>Contact Pairs**.

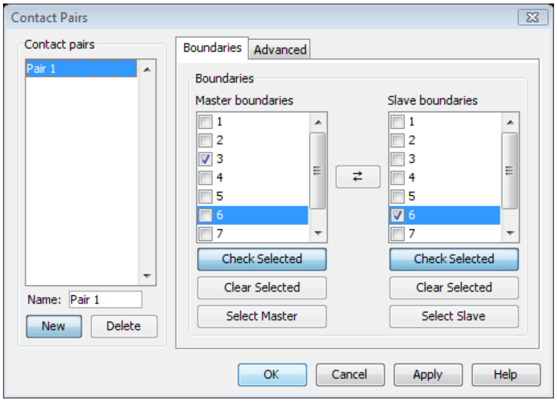
2 Click the **New** button.

3 Select Boundary 3 from the **Master boundaries** list.

4 Click the **Check Selected** button below the **Master boundaries** list.

5 Select Boundary 6 from the **Slave boundaries** list.

6 Click the **Check Selected** button below the **Slave boundaries** list.



7 Click **OK** to close the **Contact Pairs** dialog box.

Boundary Settings

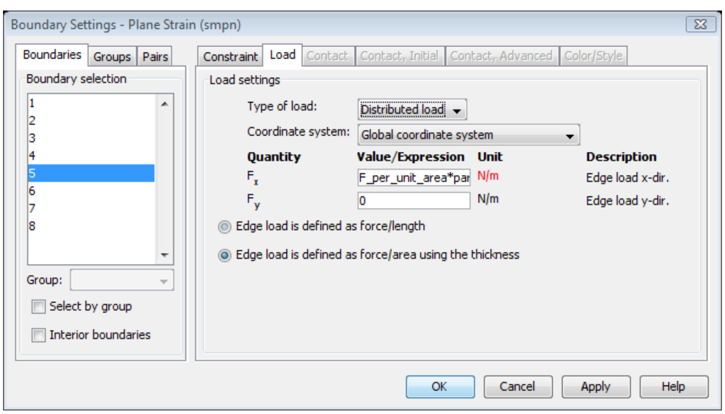
1 Select **Physics>Boundary settings**.

2 From the **Boundary selection** list select Boundary 5.

3 Click the **Load** tab.

4 Click the **Edge load is defined as force/length** button.

5 In the **F$_x$** edit field type `F_per_unit_area*para-K_per_length*u`.



6 Click the **Constraint** tab.

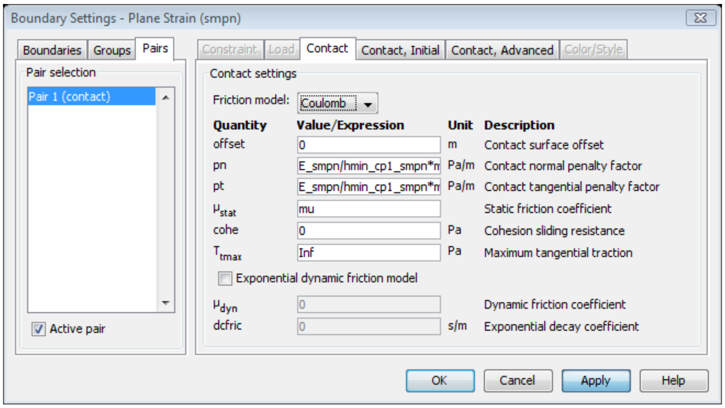
7 Select Boundary 2 from the **Boundary selection** list.

8 Select **Fixed** from the **Constraint condition** list.

9 Click the **Pairs** tab, then select **Pair 1 (contact)** from the **Pair selection** list.

10 On the **Contact** page, select **Coulomb** from the **Friction model** list.

11 In the **Static friction coefficient** edit field type `mu`.



12 Click **OK**.

COMPUTING THE SOLUTION

1 Select **Solve>Solver Parameters**.

2 Select **Parametric** from the **Analysis** list in the **Solver Parameters** dialog box.

3 Enter `para` in the **Parameter name** edit field on the **General** page.

4 In the **Parameter values** edit field type `0 0.03 0.2 0.4 0.8 1`.

5 Click the **Stationary** tab.

6 Enter `1e-13` in the **Relative tolerance** edit field.

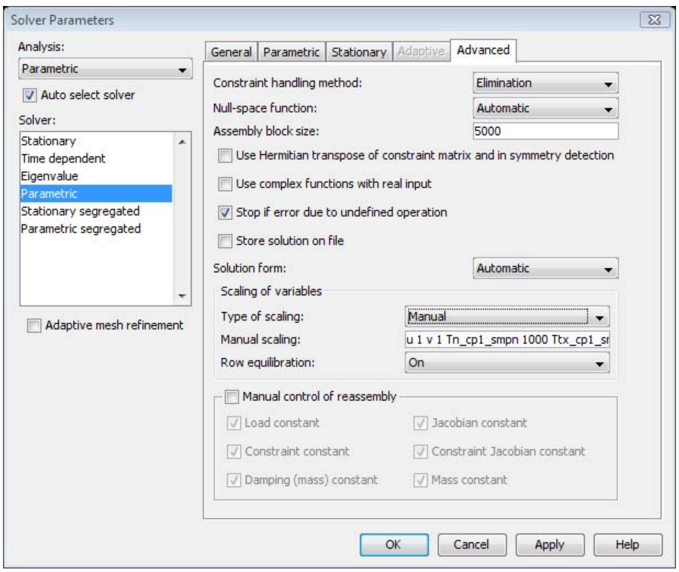
This is necessary to ensure convergence; see the section "Solver Settings for Contact Modeling" on page 129 of the *Structural Mechanics Module User's Guide* for a description of the calculation of this tolerance.

7 Click the **Advanced** tab.

8 Select **Manual** from the **Type of scaling** list in the **Scaling of variables** area.

9 In the **Manual scaling** edit field type `u 1 v 1 Tn_cp1_smpn 1000 Ttx_cp1_smpn 100 Tty_cp1_smpn 10`

This is an example of the so-called property-pair syntax, where you first enter a variable and then the corresponding scaling factor, and so on. The first two variables in the list, u and v, are the dependent variables, that is, the displacements. The rest of the variables are related to contact modeling; see the subsection “Contact Modeling” on page 186 of the *Structural Mechanics Module User’s Guide* for a description of these variables.



10 Click **OK**.

11 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To visualize the displacement of the wedge, proceed with the following instructions:

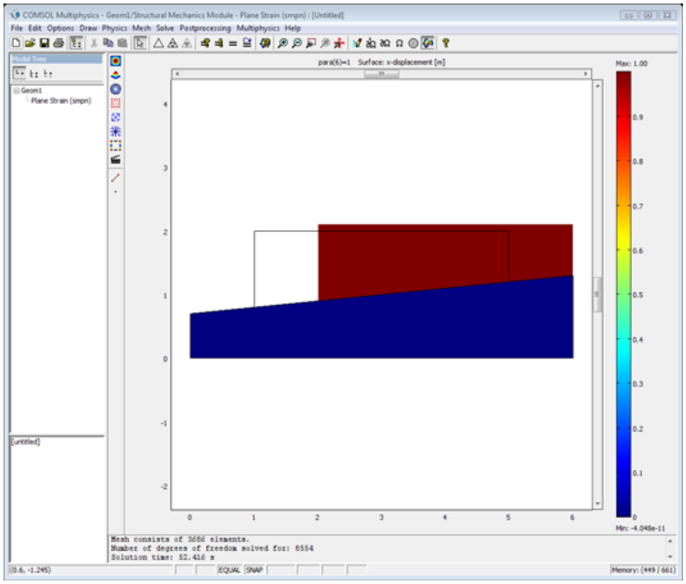
1 Click the **Plot Parameters** button.

2 Click the **Surface** tab.

3 Make sure that the **Surface plot** check box is selected, then select **x-displacement** from the **Predefined quantities** list.

4 Click **OK**.

You should now see the following plot in the main user interface window.



2D Cylinder Roller Contact

Introduction

Consider an infinitely long steel cylinder resting on a flat aluminum foundation, where both structures are elastic. The cylinder is subjected to a point load along its top. The objective of this study is to find the contact pressure distribution and the length of contact between the foundation and the cylinder. An analytical solution exists, and this model includes a comparison against the COMSOL Multiphysics solution. This model is based on a NAFEMS benchmark (see Ref. 1).

Model Definition

This is clearly a plane strain problem and the Plane Strain application mode from the Structural Mechanics Module is thus a suitable (two-dimensional) application mode. The geometry is reduced to half of the geometry at the vertical symmetry axis due to symmetry reasons.

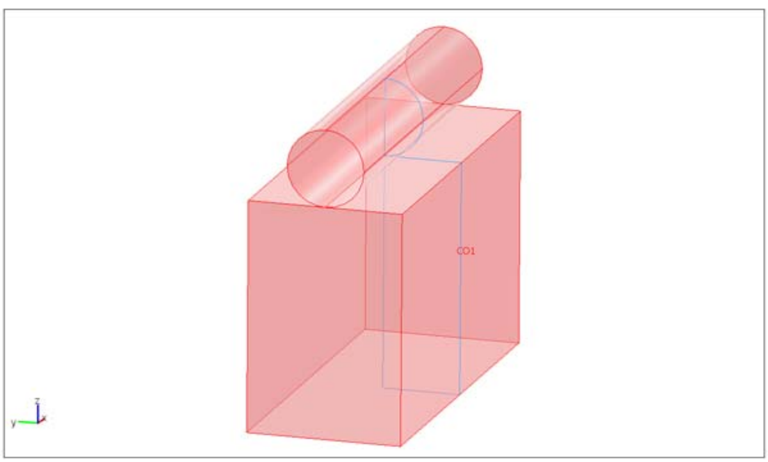


Figure 7-5: Geometry in model

The cylinder is subjected to a point load along its top with an intensity of 35 kN. Both the cylinder and block material are elastic, homogeneous, and isotropic.

The contact modeling method in this example does not include friction as described in Ref. 1. This model uses a contact pair, which is a straightforward way to implement a contact problem using the Structural Mechanics Module.

Results and Discussion

The deformed shape and the Von Mises stress distribution are depicted in Figure 7-6.

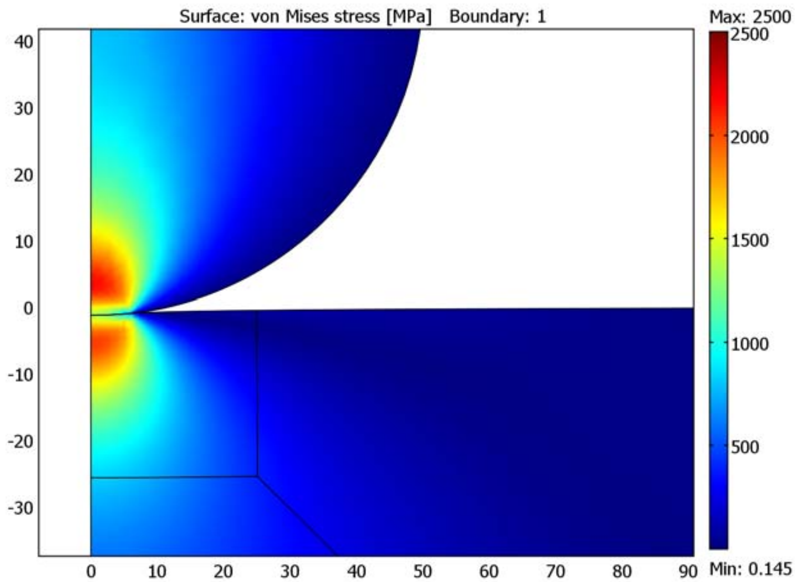


Figure 7-6: Deformation and Von Mises stress at the contact area.

The analytical solution for the contact pressure as a function of the x-coordinate is given by

$$P = \sqrt{\frac{F_n E'}{2\pi R'} \times \left(1 - \left(\frac{x}{a}\right)^2\right)}$$

$$a = \sqrt{\frac{8 F_n R'}{\pi E'}}$$

where F_n is the applied load (load/length), E' the combined elasticity modulus, R' the combined radius, and x is the x-axis coordinate.

The combined Young's modulus and radius are defined as:

$$E' = \frac{2E_1E_2}{E_2(1-\nu_1^2)+E_1(1-\nu_2^2)}$$

$$R' = \lim_{R_2 \to \infty} \frac{R_1R_2}{R_1+R_2} = R_1$$

In these equations, E_1 and E_2 are Young's modulus of the roller and the block, respectively, and R_1 is the radius of the roller.

This gives a contact length of 6.21 mm and a maximum contact pressure of 3585 MPa.

The contact pressure along the contact area for both the analytical and the COMSOL Multiphysics solution are depicted in Figure 7-7. The COMSOL Multiphysics solution is the solid line, the analytical result is the dashed line.

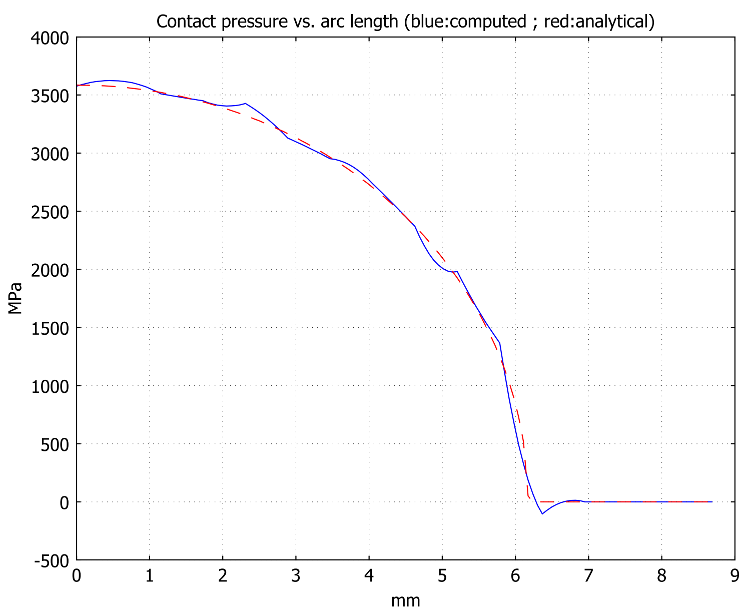


Figure 7-7: Analytical pressure distribution and COMSOL Multiphysics solution (dashed).

The maximum contact pressure obtained with COMSOL Multiphysics is 3581 MPa, which is in good agreement with the analytical solution.

The maximum contact length (equivalent to a null contact pressure) is evaluated at 6.31 mm.

References

1. Konter A.W.A., *Advanced finite element contact benchmarks*, NAFEMS, 2006

2. Crisfield M. A., *Non-linear Finite Element Analysis of Solids and Structures, volume 2: Advanced Topics*, John Wiley & Sons Ltd., England, 1997.

Modeling Using COMSOL Multiphysics

The Structural Mechanics Module supports contact boundary conditions using contact pairs. The contact pair is defined by a master (contacting) boundary and a slave (contacted) boundary. The contact boundary pair comprises a flat boundary and a curved boundary. The flat boundary is defined as the master boundary and the curved boundary as the slave boundary.

Because contact is only present in a small area, a local mesh refinement is required. Due to the geometry shape, you use a free mesh for the cylinder domain and a mapped mesh for the aluminum block. The block geometry requires some modification in order to set up a refined mesh area.

This example assumes a nominal thickness (thickness = 1).

The selected length unit in this model is mm, so the MPa unit system, which is based on mm as the length unit, is a suitable unit system.

Model Library path: `Structural_Mechanics_Module/Contact_and_Friction/cylinder_roller_contact`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 On the **New** page, select **2D** from the **Space dimension** list.

2 Expand to the next submenu by clicking on **Structural Mechanics Module> Plane Strain>Static analysis**.

3 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu select **Constants** and enter the following constant names, expressions, and descriptions (optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
E1	70e3[MPa]	Block Young's modulus
E2	210e3[MPa]	Cylinder Young's modulus
nu	0.3	Poisson's ratio
Fn	35[kN]	External load
E_star	2*E1*E2/((E1+E2)*(1-nu^2))	Combined Young's modulus
R	50[mm]	Combined radius
th	1[mm]	Thickness
lc	10[mm]	Estimated contact length
a	sqrt(8*Fn*R/(pi*E_star*th))	Contact length
pmax	sqrt(Fn*E_star/(2*pi*R*th))	Maximum contact pressure

GEOMETRY MODELING

1 In the **Draw** menu select **Use Assembly**.

1 Shift-click the **Ellipse/Circle (Centered)** button on the Draw toolbar, then set the **Radius** to 50 and the position of the center to (0, 50). Click **OK**.

2 Shift-click the **Rectangle/Square** button on the Draw toolbar and set both **Width** and **Height** to 200. Set the corner position to (–100, –200). Click **OK**.

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

4 Draw a second rectangle, this one of width 100 and height 300 with the lower left corner at (–100, –200).

5 Click the **Create Composite Object** button on the Draw toolbar and type the expression (C1+R1)-R2 in the **Set formula** edit field. Click **OK**.

6 Choose **Specify Objects>Square** from the **Draw** menu to create two squares with the following specifications:

PROPERTY	SQUARE 1	SQUARE 2
Width	100	25

Base	Corner	Corner
x	0	0
y	-100	-25

7 Choose **Specify Objects>Line** from the **Draw** menu and create a line between grid point (25, –25) and grid point (100, –100).

The mesh needs to be refined in the contact area. To do this in an efficient way, divide the curved boundary into two separate boundaries. To achieve this, add a point along the curved boundary.

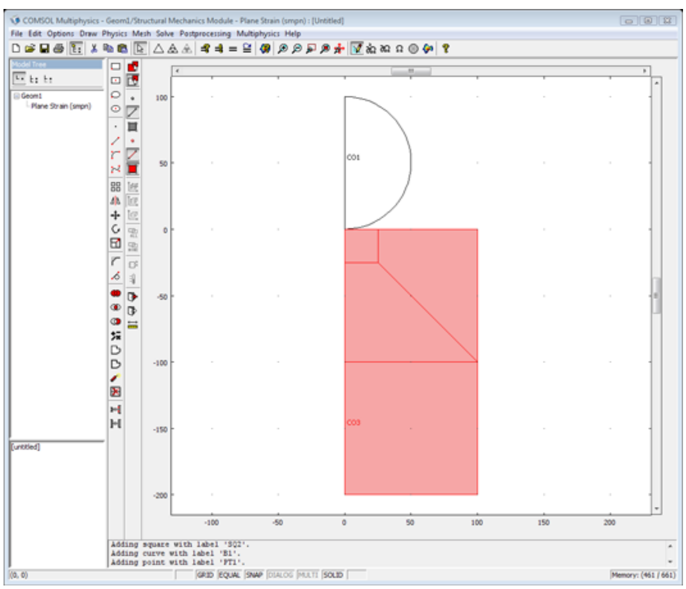
8 Draw a **Point** at the origin (0, 0).

9 Click the **Rotate** button on the Draw toolbar, then set the **Rotation angle** to 10 degrees and the *y*-coordinate of the **Center point** to 50.

10 Select all domains (press Ctrl+A), then click the **Split Object** button on the Draw toolbar.

11 Select all the geometry objects of the top part (CO3 and PT2), then click the **Coerce to Solid** button on the Draw toolbar.

12 Select all the geometry objects of the bottom part (CO2, CO4, CO5, and B2), then click the **Coerce to Solid** button.



PHYSICS SETTINGS

Model Settings

1 In the **Physics** menu, go to **Model Settings** and select **MPa** from the **Base unit system** list.

2 Click **OK**.

Subdomain Settings

1 Choose **Physics>Subdomain Settings** to specify the subdomain settings for the Plane Strain application mode according to the following table:

SETTINGS	SUBDOMAINS 1–4	SUBDOMAIN 5
E	E1	E2
ν	nu	nu

The nominal thickness is used; make sure that **thickness** is set to 1 for all subdomains.

2 Click **OK**.

Boundary Conditions

1 Choose **Physics>Boundary Settings** to open the **Boundary Settings** dialog box; then specify the following constraint conditions:

SETTINGS	BOUNDARIES 1, 3, 5, 13	BOUNDARY 2
Constraint condition	Symmetry plane	Fixed

The other boundaries are free, which is the default boundary condition.

2 Click **OK**.

Point Settings

The external load is a point load. Due to the symmetry, apply one half of the total force on the studied geometry:

1 From the **Physics** menu select **Point Settings**.

2 In the **Point Settings** dialog box select Point 11.

3 Click the **Load** tab, and then type `-Fn/2` in the **F_y** edit field.

4 Click **OK**.

Contact Pairs

Define a contact pair for Boundaries 7 and 14. A general advice is to set the curved surface as slave (contacting) boundary and the straight surface as master (contacted) boundary. The top boundary (number 7) is set to slave boundary and the bottom boundary (number 14) as master boundary in this model. The steel block is also the correct master surface because it has a high stiffness.

1 From the **Physics** menu select **Contact Pairs**.

2 In the **Contact Pairs** dialog box, click **New**.

3 In the **Master boundaries** list, select Boundary 7.

4 Click the **Check Selected** button below the **Master boundaries** list.

5 In the **Slave boundaries** list select Boundary 14.

6 Click the **Check Selected** button below the **Slave boundaries** list.

7 Click **OK**.

   The next step is to specify the contact parameters in the **Boundary Settings** dialog box.

8 In the **Boundary Settings** dialog box, click the **Pairs** tab, and select **Pair 1 (contact)**.

Note: To reduce the number of iteration steps and improve convergence, it is good practice to set an initial contact pressure as close to the solution as possible. A good approximation is to use the value of the external pressure, in this case the external point load divided by an estimated contact length and the thickness. In this model you need to set an initial contact pressure to make the model stable for the initial conditions.

---

9 On the **Contact, Initial** page, type `(Fn/2)/(lc*th)` in the **Contact pressure** edit field (the division by 2 is to account for the symmetry).

10 Click **OK**.

MESH GENERATION

You can achieve an accurate contact pressure by refining the mesh around the contact zone. A general recommendation is to have at least 10 contacting nodes along the slave contact boundary. Apply an element size of 0.59 mm for the cylinder and 1.5 mm for the block. Due to the geometry shape, use a free mesh (triangular elements) for the cylinder and a mapped mesh (quadrilateral elements) for the block.

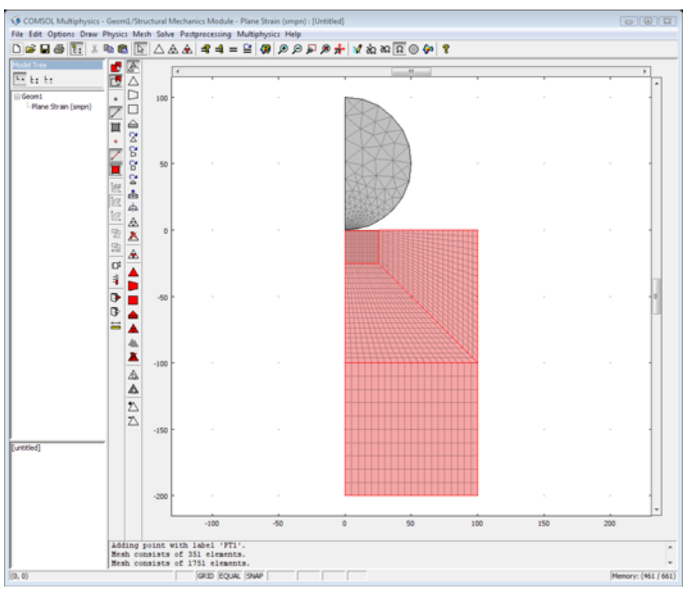
1 Open the **Free Mesh Parameters** dialog box by selecting **Free Mesh Parameters** from the **Mesh** menu.

2 Click the **Boundary** tab. Select Boundary 14, then type `0.59` in the **Maximum element size** edit field.

3 Click the **Subdomain** tab, then select Subdomain 5. Click **OK**.
4 Click the **Mesh Selected (Free)** button on the Mesh toolbar.

   In the message log you can see that the free mesh is composed of around 360 elements.
5 Open the **Mapped Mesh Parameters** dialog box by selecting **Mapped Mesh Parameters** from the **Mesh** menu.
6 On the **Boundary** page, select Boundaries 3, 5, and 7. Then select **Constrained edge element distribution** and type `20` in the **Number of edge elements** edit field.
7 On the **Boundary** page, select Boundary 1. Then select the **Constrained edge element distribution** and type `10` in the **Number of edge elements** edit field. Click **Apply**.
8 Go to **Subdomain** page and select Subdomains 1, 2, 3, and 4.
9 Click the **Mesh Selected** button. Click **OK**.

The information in the message log shows that the total mesh consists of approximately 1750 elements (roughly 360 elements for the free mesh and 1400 elements for the mapped mesh).



COMPUTING THE SOLUTION

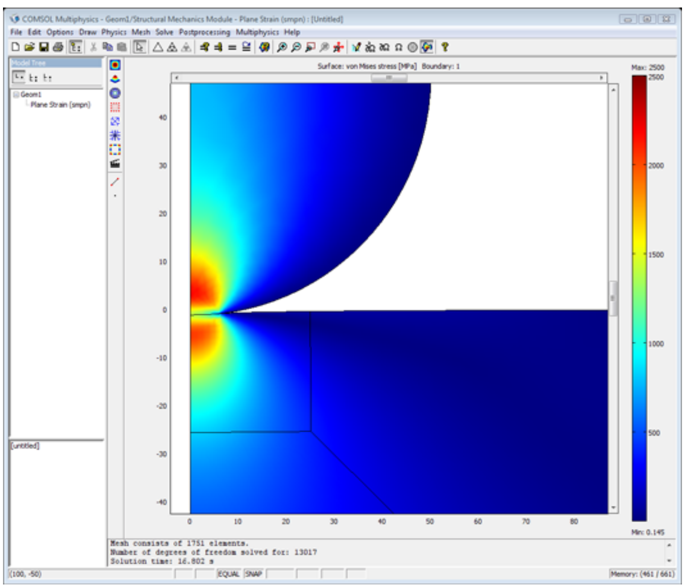
Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows the von Mises stress. Because of the point-load constraint, a stress concentration develops on the top of the cylinder. Due to the automatic plot-range settings, the stress around the contact surface looks almost identical.

1. Click the **Plot Parameters** button on the Main toolbar.
2. On the **General** page, select the **Surface** and **Boundary** check boxes and clear the **Geometry edges** check box in the **Plot type** area.
3. On the **Surface** page, click the **Range** button. In the **Color Range** dialog box clear the **Auto** check box, then type `2500` in the **Max** edit field. Click **OK**.
4. On the **Boundary** page, type `1` in the **Expression** edit field on the **Boundary Data** tab. Click the **Uniform color** option button in the **Boundary color** area, then click the **Color** button and select black. Click **OK**.
5. Click **OK**.

Zoom in on the contact area to get the following plot:



The instructions below show how to create a line plot to compare the analytical contact pressure distribution with the contact pressure calculated by the model.

1 Choose **Options>Expressions>Boundary Expressions** and select Boundary 14. Enter the following constant names and expressions:

NAME	EXPRESSION
theta	atan2(y-50,x)+pi/2
x_local	sqrt(x^2+(y-50)^2)*theta
p_analytical	pmax*sqrt(1-(x_local/a)^2)

2 Click **OK**.

3 In the **Solve** menu, select **Update Model**.

You can now plot the analytical expression of the contact pressure.

4 Choose **Postprocessing>Domain Plot Parameters**.

5 Click the **Line/Extrusion** tab, then from the **Predefined quantities** list select **Contact pressure, contact Pair 1**.

6 Select Boundary 14 (the slave boundary), then click **Apply** to generate a plot of the contact pressure calculated in the model.

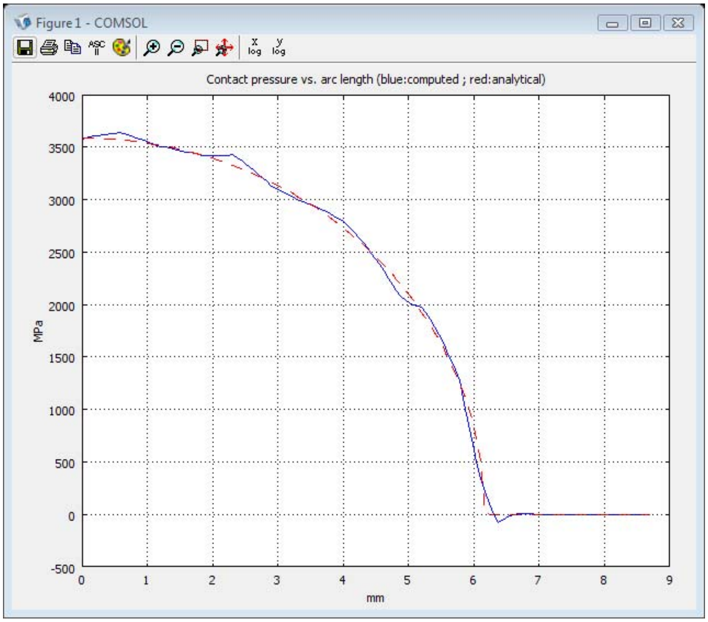
7 In the **Expression** edit field type `p_analytical`.

8 Click the **Line Settings** button, then select **Color** in the **Line color** list, and **Dashed line** in the **Line style** list. Click **OK**.

9 Click the **General** tab, then select **Keep current plot**.

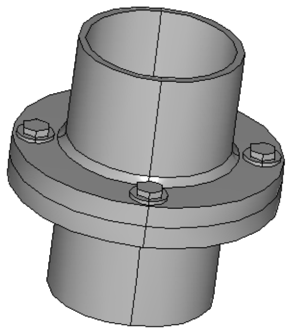
10 Click the **Title/Axis** button and type `Contact pressure vs. arc length (blue: calculated, red: analytical)` in the **Title** edit field. In the **First axis label** edit field type `mm` and in the **Second axis label** edit field type `MPa`. Click **OK**.

11 Click **OK** in **Domain Plot Parameters** dialog box to generate the plot of the analytical expression for the contact pressure.



Tube Connection

A tube connection consisting of a flange with four prestressed bolts is subjected to tensile forces. A sketch of the connection is shown below.



Model Definition

The tube is made of steel and has an outer diameter of 220 millimeters and an inner diameter of 200 millimeters. The flange has a diameter of 360 millimeters and it is 30 millimeters thick. The connection consists of four prestressed M24 bolts. The bolts are prestressed to 80% of the yield strength. The tensile force in the tube varies from 0 to 500 kN.

To compute the influence of the tensile force on the stress level in the bolt, the model includes a parametric analysis. Because of symmetry in both load and geometry, you only need to analyze one eighth of one of the flanges. The geometry has been created in the CAD software SolidWorks and is available both as an IGES file and as a SolidWorks file. The model exists in two different versions depending how the geometry is imported. The IGES version (`tube_connection_igs.mph`) is created from the IGES file through the IGES file import. Read more about IGES import in the section Importing 3D IGES Files in the *CAD Import Module User's guide*. The SolidWorks version (`tube_connection_sw.mph`) is created from the SolidWorks live connection. Read more about the SolidWorks interface in the section "SolidWorks Live Connection" on page 30 in the *CAD Import Module User's guide*.

A contact pair between the bottom surface of the flange and the top surface of an additional fixed solid models the contact. To represent the bolt, the model uses a distributed linear boundary condition on a surface of the size of the washer.

Note: This model requires both the Structural Mechanics Module and the CAD Import Module.

Modeling in COMSOL Multiphysics

Two application modes for 3D solid stress-strain analysis are used to solve the problem. One application mode models the initial prestress problem of calculating the deformation from the prestressed bolts. The other is used for solving the parametric problem including the tensile force, using the solution from the prestress application mode to set up the correct boundary conditions.

Results and Discussion

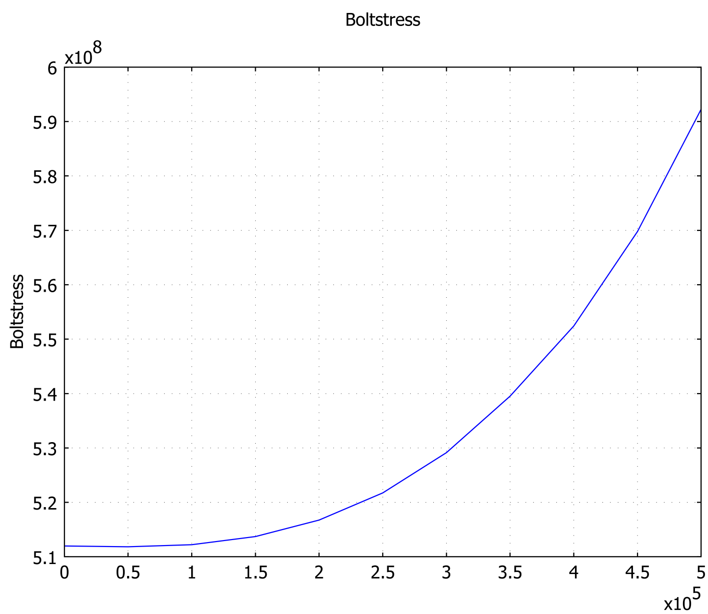


Figure 7-8: The bolt stress as a function of the tensile force.

For a tensile force below 250 kN the prestressed connection works well, the bolt stress increasing less than 2%. In the region 250 kN–500 kN the bolt stress increases by more than 13% due to the nonlinear effects in the connection.

Model Library path: `Structural_Mechanics_Module/Contact_and_Friction/tube_connection_igs`

Model Library path: `Structural_Mechanics_Module/Contact_and_Friction/tube_connection_sw`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Click the **New** button or start COMSOL Multiphysics to open the **Model Navigator**.
2. Select **3D** in the **Space dimension** list.
3. In the list of application modes, open the **Structural Mechanics Module** folder and then **Solid, Stress-Strain**. Select the **Static analysis**.
4. Click **OK**.

OPTIONS AND SETTINGS

1. From the **Options** menu, select **Constants**.
2. Enter the following constant names, expressions, and (optionally) descriptions. When finished, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`Abolt`	`pi/4*24e-3^2[m^2]`	Bolt area
`Atube`	`pi/4*(0.22^2-0.2^2)[m^2]`	Tube area
`Force`	`0[N]`	Tensile force in the tube
`Sboltini`	`800*0.8*0.8*1e6[N/m^2]`	Prestress in the bolt
`Awash`	`pi/4*(0.05^2-0.025^2)[m^2]`	Washer area
`thickness`	`0.03[m]`	Flange thickness

NAME	EXPRESSION	DESCRIPTION
Emod	2e11[N/m^2]	Modulus of elasticity of the bolt
Kbolt	Emod*Abolt/thickness/Awash	Distributed stiffness of the bolt (N/m^3)
Fbolt	Sboltini*Abolt/Awash	Distributed prestress from the bolt (N/m^2)

GEOMETRY MODELING

There are two different ways to create the geometry:

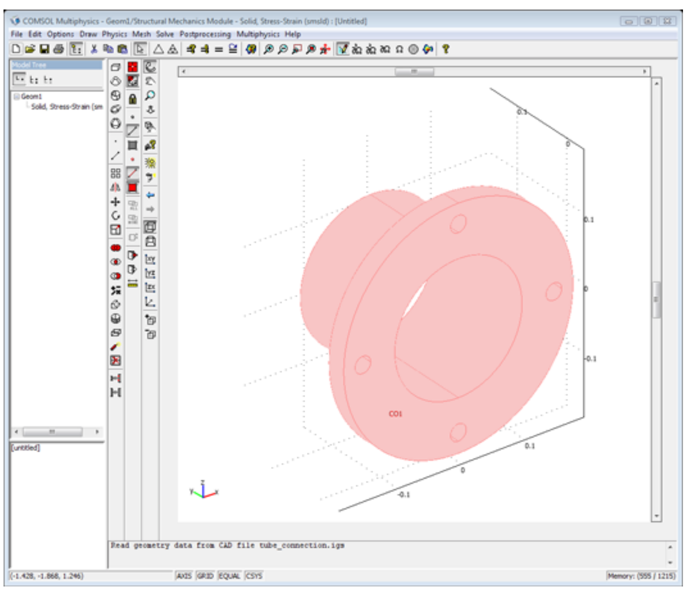
- From an IGES file, through the IGES file import.
- From a SolidWorks file, through the SolidWorks live connection.

IGES File Import

1 On the **File** menu, select **Import>CAD Data From File** to open the **Import CAD Data From File** dialog box.

2 Open the IGES file `tube_connection.igs` located in the `models\Structural_Mechanics_Module\Contact_and_Friction` folder in the COMSOL installation directory.

3 Click **Import**.



Due to symmetry, you can reduce the geometry to one eighth of the full geometry. To do so, create a 45-degree wedge and then intersect it with the imported geometry:

1 Select **Work-Plane Settings** from the **Draw** menu.
2 Click **OK** in the **Work-Plane Settings** dialog box as you will use the default settings.
3 Click the **Zoom Extents** toolbar button.
4 Draw a rectangle with opposite corners at (0, –0.1) and (0.2, 0.2).
5 Select **Revolve** from the **Draw** menu and enter `45` as **α2** in the **Revolve** dialog box. Click **OK**.
6 Press Ctrl+A to select all objects and then click the **Intersection** button on the Draw toolbar to create an eighth.

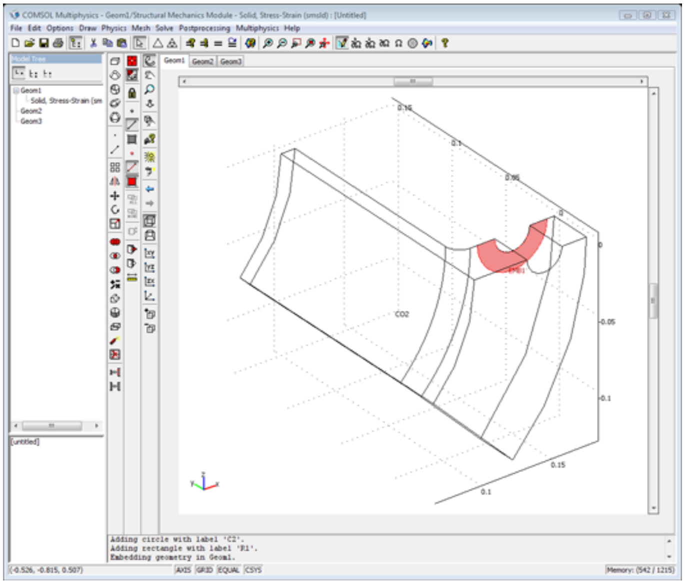
Model a washer by creating a boundary around the hole. The distributed forces from the bolt are applied on this boundary.

1 Select **Work-Plane Settings** from the **Draw** menu.
2 Click the **Add** button, select the **z-x** option in the **Plane** area and click **OK**.
3 Set grid settings in Geom 3 according to the following table:

GRID	
x spacing	5e-3
Extra x	
y spacing	5e-3
Extra y	

4 Click the **Zoom Extents** button on the Main toolbar to see the geometry.
5 Press the Shift key and click the **Ellipse/Circle (Centered)** button on the Draw toolbar to create a centered circle with radius 0.025 and center at (0, 0.15).
6 Draw another centered circle with radius 0.0125 and center at (0, 0.15).
7 Select both circles and click the **Difference** button to create a washer.
8 Draw a rectangle with opposite corners at (0, 0.12) and (0.03, 0.18).
9 Click the **Create Composite Object** button on the Draw toolbar, type `CO1-R1` in the **Set formula** edit field, and click **OK** to close the dialog box.

10 Select **Embed** from the **Draw** menu, then click **OK**.



Create an additional solid below the bottom flange (this solid is needed for the contact modeling).

1 Select **Work-Plane Settings** from the **Draw** menu.

2 Click the **Add** button, select the **x-y** option in the **Plane** area and click **OK**.

3 Set grid settings in Geom4 according to the following table:

GRID	
x spacing	0.01
Extra x	
y spacing	0.01
Extra y	

4 Draw a rectangle with opposite corners at (0.09, –0.04) and (0.19, –0.03). To do so, press Shift and click the **Rectangle/Square** button on the Draw toolbar. In the **Rectangle** dialog box, enter (0.09, –0.04) as the coordinates for lower-left corner and a width and height of 0.10 and 0.01, respectively.

5 Select **Revolve** from the **Draw** menu and enter 45 as **α2** in the **Revolve** dialog box. Click **OK**.

You need to work in assembly mode to model the contact between the two initially connecting solids. The embedded washer surface must be a part of a solid to be included in the assembly between solids.

1. Select the upper solid part, CO2, and the embedded surface, EMB1.
2. Click the **Coerce to Solid** button on the Draw toolbar to make the washer a part of the solid.
3. Select **Use Assembly** from the **Draw** menu to leave draw mode in assembly mode.

SolidWorks Live Connection

Another way to create your geometry is to import it directly from SolidWorks. To follow these instructions you need to have SolidWorks installed on your computer.

1. Start SolidWorks and open the SolidWorks file `tube_connection.SLDPRT` located in the `models/Structural_Mechanics_Module/Contact_and_Friction` in the COMSOL installation directory.
2. Switch to COMSOL and select **File>SolidWorks Connection>Initialize**.

Create an additional solid below the bottom flange, this solid is needed for the contact modeling.

1. Select **Work-Plane Settings** from the **Draw** menu.
2. Click **OK** in the **Work-Plane Settings** dialog box as you will use the default settings.
3. Set grid settings in Geom2 according to the following table:

GRID	
x spacing	`0.01`
Extra x	
y spacing	`0.01`
Extra y	

4. Draw a rectangle with opposite corners at (0.09, –0.04) and (0.19, –0.03).
5. Select **Revolve** from the **Draw** menu and enter `45` as **α2** in the **Revolve** dialog box. Click **OK**.

   You need to work in assembly mode to model the contact between the two initially connecting solids.
6. Select **Use Assembly** from the **Draw** menu to leave draw mode in assembly mode.

PHYSICS SETTINGS

Boundary Settings

In the first application mode you compute the initial deformation from the prestressed bolt. You then use a second Solid Stress-Strain application mode for the parametric analysis where the tensile force in the tube is increased. You need the solution from the first application mode to set up the boundary conditions for the parametric analysis.

1 Select **Model Navigator** from the **Multiphysics** menu to open the **Model Navigator**.

2 Select **Structural Mechanics Module>Solid, Stress-Strain** from the application mode tree and click the **Add** button to add a second solid stress-strain application mode to your 3D geometry. Click **OK** to close the **Model Navigator**.

   To model the contact, use contact pairs. Start by creating two contact pairs, one for each application mode. Make the weaker boundary the slave and the stiffer one the master boundary.

3 Select **Physics>Contact Pairs** to open the **Contact Pairs** dialog box.

4 Click the **New** button to create a contact pair.

5 Select Boundary 4 in the **Master boundaries** list.

6 Select the check box in front of **4** in the **Master boundaries** list.

7 Select Boundary 9 in the **Slave boundaries** list.

8 Select the check box in front of **9** in the **Slave boundaries** list.

9 Click the **New** button to create an additional contact pair.

10 Select the same boundaries as master and slave boundaries as you made for the first pair.

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

12 Select **1 Geom1: Solid, Stress-Strain (smsld)** from the **Multiphysics** menu.

13 Select **Physics>Boundary Settings** and clear the **Active pair** check box for **Pair 2**.

14 Select **2Geom1: Solid, Stress-Strain (smsld2)** from the **Multiphysics** menu.

15 Select **Physics>Boundary Settings** and clear the **Active pair** check box for **Pair 1**.

16 Select **1 Geom1: Solid, Stress-Strain (smsld)** from the **Multiphysics** menu.

17 Select **Physics>Boundary Settings** and specify the boundary and contact-pair settings according to the following tables:

	BOUNDARIES 8, 14, 18		BOUNDARY 15	
Page	Constraint		Load	
	Symmetry plane		F_y	`-Fbolt`

	PAIR 1		PAIR 2
Page	Contact		
	P_n	`E_smsld/h*min(1e-3*5^auglagiter,100)`	
Page	Contact, Initial		
	T_n	`1e7`	

The `auglagiter` variable is the iteration number in the augmented Lagrange solver. It makes the penalty parameter soft at the beginning (to help the solver get started) and then gradually makes it stiffer (to speed up convergence).

18 Select **2 Geom1: Solid, Stress-Strain (smsld2)** from the **Multiphysics** menu.

19 Select **Physics>Boundary Settings** and specify the boundary and contact pair settings according to the following tables:

	BOUNDARIES 8, 14, 18		BOUNDARY 10		BOUNDARY 15	
Page	Constraint		Load		Load	
	Symmetry plane		F_y	`Force/Atube`	F_y	`-Fbolt-Kbolt*(v2-v)`

	PAIR 2		PAIR 1
Page	Contact		
	P_n	`E_smsld2/h*min(1e-3*5^auglagiter,100)`	
Page	Contact, Initial		
	T_n	`Tn_cp1_smsld`	

20 For enabling plotting of the bolt stress as a function of the applied tensile force an integration coupling variable is created, representing the integral of the force on the washer boundary.

21 Select **Options>Integration Coupling Variables>Boundary Variables** to open the **Boundary Integration Variables** dialog box.

22 Select Boundary 15 in the **Boundary selection** list.

23 In the **Name** column type `Boltstress`.

24 In the **Expression** column type `(Fbolt+Kbolt*(v2-v))/(Abolt/2)`.

25 Leave **Global destination** enabled and click **OK** to close the dialog box.

Subdomain Settings

The material properties will be defined using the materials library and are identical for both application modes. Specify the extra solid (added for the contact modeling) to be fixed.

1 Select **1 Geom1: Solid, Stress-Strain (smsld)** from the **Multiphysics** menu to set the subdomain parameters for the first application mode.

2 Select **Physics>Subdomain Settings**.

3 Select Subdomain 2 from the subdomain list. 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 and click **OK** to close the **Materials/Coefficients Library** dialog box.

5 Select Subdomain 1 from the subdomain list. Click the **Constraint** page and select **Fixed** from the **Constraint condition** list.

6 Click **OK** to close the **Subdomain Settings** dialog box.

Do the same for the other application mode:

7 Select **2 Geom1: Solid, Stress-Strain (smsld2)** from the **Multiphysics** menu to set the subdomain parameters for the second application mode.

8 Select **Physics>Subdomain Settings**.

9 Select Subdomain 2 from the subdomain list.

10 Select **Structural Steel** from the **Library material** list.

11 Select Subdomain 1 from the subdomain list. Click the **Constraint** page and select **Fixed** from the **Constraint condition** list.

12 Click **OK** to close the **Subdomain Settings** dialog box.

MESH GENERATION

The default mesh settings are used.

COMPUTING THE SOLUTION

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

2 Click the **Stationary** tab and find the **Augmented Lagrangian solver** area.

3 In the **Tolerance** edit field type `1e-5`.

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

First solve the initial problem (no tensile force, only prestress) to determine the initial displacements.

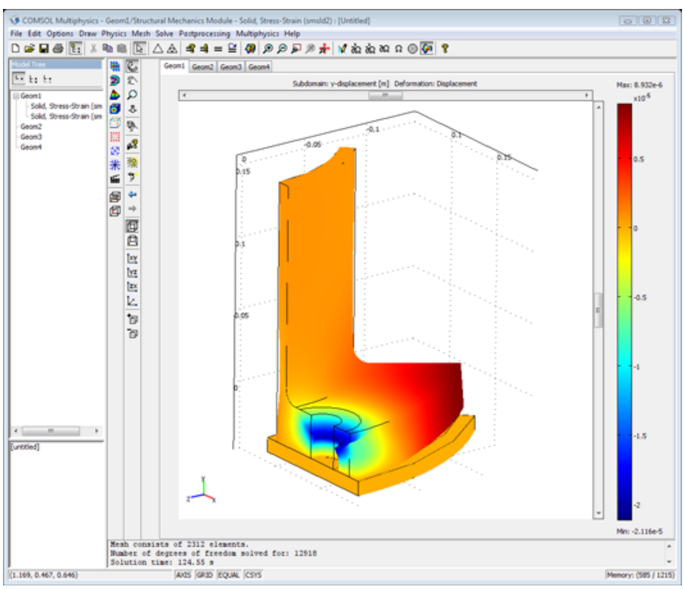
1. Select **Solver Manager** from the **Solve** menu to open the **Solver Manager** dialog box.
2. On the **Solve For** page select **Geom1: Solid, Stress-Strain (smsld)** from the **Solve for variables** list, then click **OK** to close the dialog box.
3. Click the **Solve** button on the Main toolbar to compute the solution.

You have now solved the prestress problem without any external load.

POSTPROCESSING AND VISUALIZATION

Visualize the displacement in the *y* direction and the deformation as follows:

1. Select **Plot Parameters** from the **Postprocessing** menu.
2. On the **General** page select **Frame (ref)** from the **Frame** list.
3. Clear **Slice** plot, then select **Deformed shape** plot and **Subdomain** plot.
4. Click the **Subdomain** tab.
5. From the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>y displacement**.
6. Click **OK**.



Now consider the externally applied tensile force.

COMPUTING THE SOLUTION

Switch to the parametric solver and solve for the displacements in the second application mode using the solution from the first application mode in the boundary condition and as initial condition.

1 Select **Solver Parameters** from the **Solve** menu to open the **Solver Parameters** dialog box.
2 Select **Parametric** in the **Analysis** list.
3 On the **General** page type `Force` in the **Parameter name** edit field and type `0:50e3:500e3` in the **Parameter values** edit field. Click **OK**.
4 Choose **Solve>Solver Manager**, then click the **Initial Value** tab.
5 Click the **Initial value expression evaluated using stored solution** option button in the **Initial value** area.
6 Click the **Stored solution** option button in the **Values of variables not solved for and linearization point** area.
7 Click the **Store Solution** button to save the initial solution.
8 Select **Geom1: Solid, Stress-Strain (smsld2)** from the **Solve for variables** list on the **Solve For** page, then click **OK** to close the dialog box.

Set the initial values in the second application mode to the result from the first application mode.

1 Select **2 Geom1: Solid, Stress-Strain (smsld2)** from the multiphysics menu.
2 From the **Physics** menu, choose **Subdomain Settings**.
3 Click the **Init** tab and specify the initial values according to the following table.

SUBDOMAIN	1, 2	
Page	Init	
	$u2(t_0)$	u
	$v2(t_0)$	v
	$w2(t_0)$	w

4 Click **OK**.
5 Click the **Solve** button on the Main toolbar to start the parametric solver.

POSTPROCESSING AND VISUALIZATION

To visualize the result of the parametric analysis, you can use a plot of the von Mises stress combined with a deformation plot:

1 From the **Postprocessing** menu, select **Plot Parameters**.

2 Select **5e5** in the **Parameter value** list on the **General** page.

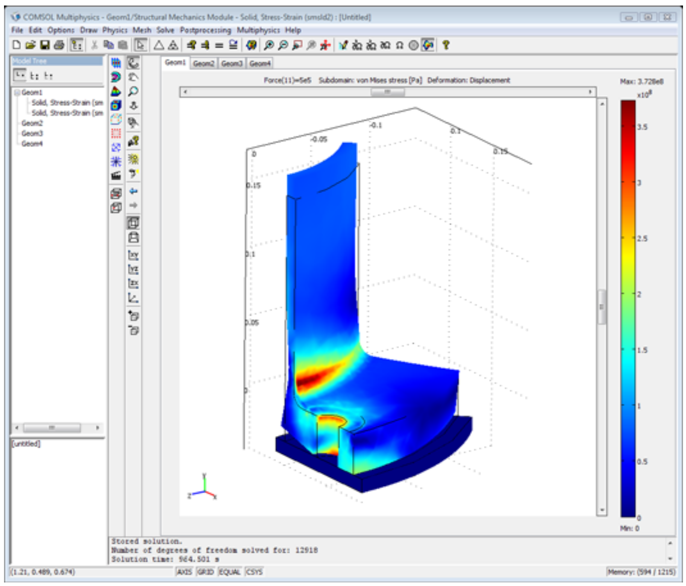
3 Click the **Deform** tab.

4 In the **Deformation data** area, select **Solid, Stress-Strain (smsld2)>Displacement** in the **Predefined quantities** list.

5 Click the **Subdomain** tab.

6 Select **Solid, Stress-Strain (smsld2)>von Mises stress** from the **Predefined quantities** list.

7 Click **OK**.



To plot the stress in the bolt as a function of the external tensile force, do the following steps:

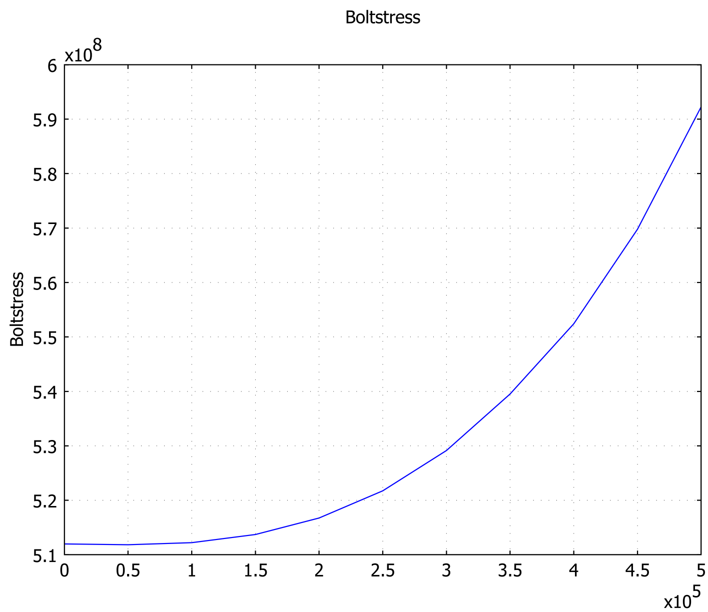
1 From the **Postprocessing** menu, choose **Global Variables Plot Parameters**.

2 Make sure that all parameter values are selected in the **Solutions to use** list.

3 Enter `Boltstress` in the **Expression** edit field and click the **>** button.

4 Select **Boltstress** from the **Quantities to plot** list.

5 Click **OK** to create the plot.



Snap Hook

Introduction

This model simulates the insertion of a snap hook in its groove. Fasteners like this are common in the automotive industry, for example, in the control panel of a car. In this case it is important to know the force that must be applied in order to place the hook in the slot but also the force needed to remove it. From a numerical point of view, this is a highly nonlinear structural analysis, mainly due to the contact interaction between the hook and the slot, but also due to the elasto-plastic constitutive law selected for the hook, and finally due to the geometrical nonlinearity originating from the large deformation.

Model Definition

Due to symmetry, you can study only half of the original snap hook geometry, this way reducing the size of the model.

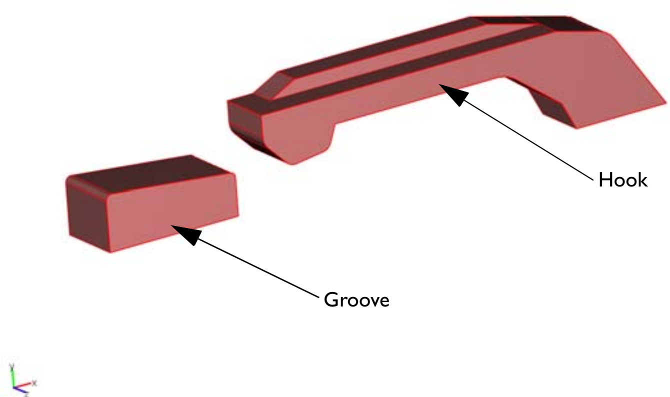


Figure 7-9: Geometry of the modeled snap hook.

MATERIAL PROPERTIES

Assume a linear elastic material for the groove with the default material properties (steel). For the hook, assume an elasto-plastic material model with isotropic hardening and a constant tangent hardening modulus, with material parameters according to the following table.

MATERIAL PARAMETER	VALUE
Young's modulus	10 GPa
Poisson's number	0.35
Yield stress	120 MPa
Isotropic tangent modulus	1.2 GPa

BOUNDARY CONDITIONS

As you can see in Figure 7-10 the applied boundary conditions are:

- A symmetry boundary condition (zero normal displacements) for boundaries in the symmetry plane.
- A fixed boundary condition (all displacement components zero) for the face of the groove where it is attached to the remainder of the geometry.
- A prescribed displacement boundary condition for the face of the hook where it is attached to the rest of the geometry. Displacement in the x direction controlled by the parametric solver while the other displacement components are zero.

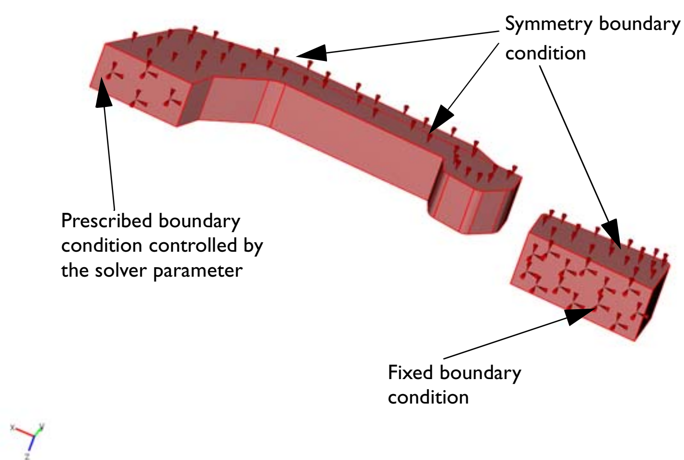


Figure 7-10: Boundary conditions in the model.

Results

The maximum effective stress levels are found at parameter step 0.8, that is, just before the hook enters the groove; see also figure Figure 7-11.

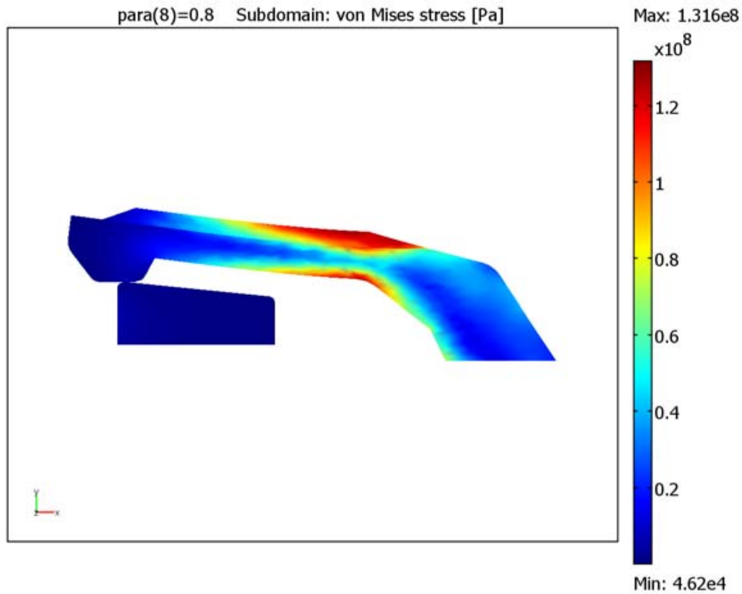


Figure 7-11: The effective stress levels in the hook just before it enters the groove.

The figure below shows the total force required for the insertion and removal of the fastener versus the parameter step. The hook is in its groove at parameter step 1. For parameter values above 1 the hook is removed from the groove.

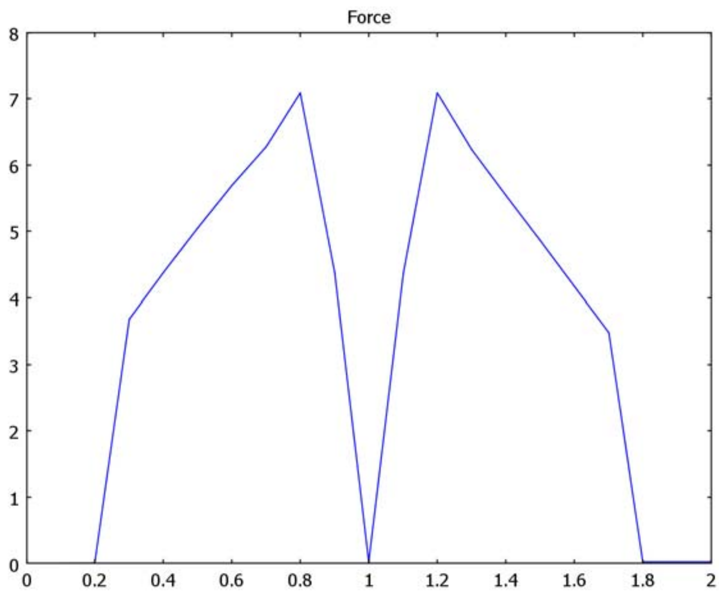


Figure 7-12: The mounting force as a function of the parameter step.

The insertion of the hook causes it to become permanently deformed. As you can see in Figure 7-13, after the hook has been removed there is a region where the plastic strains are greater than zero. This means that the hook has not returned to its original shape.

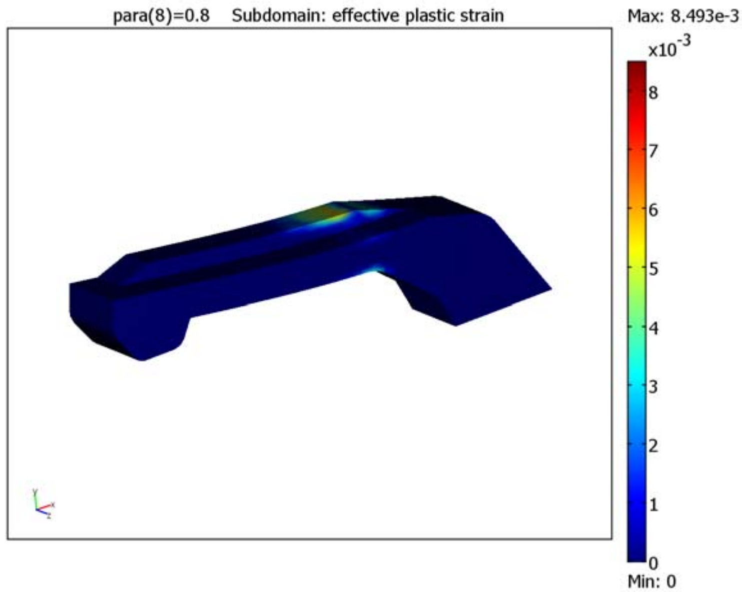


Figure 7-13: Effective plastic strain in the hook, after its removal from the groove.

Model Library path: `Structural_Mechanics_Module/Contact_and_Friction/snap_hook_fastener`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Click the **New** button or start COMSOL Multiphysics to open the **Model Navigator**.
2. Select **3D** from the **Space dimension** list.
3. Open the **Structural Mechanics Module** folder and then **Solid, Stress-Strain**. Select **Parametric analysis**.
4. From the **Element** list box select **Lagrange - Linear**.
5. Click **OK**.

GEOMETRY MODELING

The geometry for the model is available as a CAD file, and you have the option of skipping the procedure of building the geometry from scratch. If you want to create the geometry yourself, skip directly to the section "Building the Geometry Step by Step" where you find detailed instructions for the procedure.

Importing the Geometry from a CAD File

1. Choose **File>Import>CAD Data From File**.
2. In the **Look in** list browse to the folder `models\Structural_Mechanics_Module\Contact_and_Friction` in your COMSOL Multiphysics installation directory.
3. Select the file `snap_hook.mphbin`, then click **Import**.

Skip the following subsection and continue building the model at "Options and Settings" on page 305.

Building the Geometry Step by Step

First you start by creating a workplane, which you can use to draw the cross-sections of the snap hook assembly.

1. Select **Work-Plane Settings** from the **Draw** menu.
2. On the **Quick** page click the **x-y** button.
3. Click **OK**.

The view in the drawing area changes to the 2D workplane, which is **Geom2**. You can continue in the workplane by drawing the cross-section of the groove that will hold the hook.

1. Hold down the Shift key and click the **Line** button on the **Draw** toolbar.
2. In the dialog box that opens enter `-0.7 -3.2` in the **x** edit field and `1.5 1.5` in the **y** edit field.
3. Click **OK**.

4 According to Steps 1 to 3 above create three more lines based on the coordinates from the following table:

LINE	X	Y
1	`-3.2 -3.2`	`1.5 2.5`
2	`-3.2 -0.7`	`2.5 2.25`
3	`-0.7 -0.7`	`2.25 1.5`

To make the lines visible in the drawing area you can use the **Zoom Extents** toolbar button.

5 Press Ctrl+A to select all the line objects and then click the **Coerce to Solid** toolbar button.

6 Click the **Fillet/Chamfer** toolbar button.

7 In the **Fillet/Chamfer** dialog box select Vertices 2 and 4 belonging to the object CO1 and enter `0.1` in the **Radius** edit field.

8 Click **OK**.

9 Select **Draw>Extrude** to open the **Extrude** dialog box.

10 In the **Objects to extrude** list select CO2 and in the **Distance** edit field enter `1.2`.

11 Click **OK**.

Now, in a similar manner, you can create the geometry for the solid body that makes up the hook.

1 Click the **Geom2** tab above the drawing area to switch back to the 2D workplane.

2 Draw each line in the following table by first holding down the Shift key while clicking the **Line** button on the **Draw** toolbar, then entering the values in the **x** and **y** edit fields of the **Line** dialog box, followed by clicking **OK**.

LINE	X	Y
1	`0 0`	`2.5 3`
2	`0 5.75`	`3 3`
3	`5.75 6.75`	`3 2.75`
4	`6.75 7.75`	`2.75 1.25`
5	`7.75 6`	`1.25 1.25`
6	`6 5.75`	`1.25 1.75`
7	`5.75 4.75`	`1.75 2.5`
8	`4.75 1.4`	`2.5 2.5`

LINE	X	Y
9	1.4 1.25	2.5 2.1
10	1.25 0.5	2.1 2
11	0.5 0	2 2.5

3 Click **Zoom Extents** on the Main toolbar.

4 In the drawing area select the lines B1 through B11 and click the **Coerce to Solid** button on the **Draw** toolbar.

5 Click the **Fillet/Chamfer** toolbar button.

6 In the **Fillet/Chamfer** dialog box select Vertex 1 from the object CO1 and enter `0.3` in the **Radius** edit field.

7 Click **Apply**.

8 Select Vertices 7 and 11 from the object CO3 and enter `0.5` in the **Radius** edit field.

9 Click **Apply**.

10 Select Vertices 4 and 5 from the object CO1 and enter `0.2` in the **Radius** edit field.

11 Click **OK**.

12 Select **Draw>Extrude** to open the **Extrude** dialog box.

13 In the **Objects to extrude** list select CO3 and in the **Distance** edit field enter `1`.

14 Click **OK**.

As the last step you can create a stiffening rib on the top of the hook, and finally scale the geometry.

1 Click the **Geom2** tab above the drawing area to switch back to the 2D workplane.

2 Create the four lines from the following table by first holding down the Shift key while clicking the **Line** button on the **Draw** toolbar, then entering the values in the **x** and **y** edit fields of the **Line** dialog box, followed by clicking **OK**.

LINE	X	Y
1	0.5 1	3 3.25
2	1 4.75	3.25 3.25
3	4.75 5.75	3.25 3
4	5.75 0.5	3 3

3 In the drawing area select the lines B1 through B4 and click the **Coerce to Solid** button on the **Draw** toolbar.

4 Select **Draw>Extrude** to open the **Extrude** dialog box.

5 In the **Objects to extrude** list select CO1 and in the **Distance** edit field enter `0.5`.

6 Click **OK**.

The **Geom1** tab becomes active again with all three 3D objects visible.

7 Select all geometry objects by pressing Ctrl+A and click the **Scale** button on the **Draw** toolbar.

8 In the **Scale factor area**, enter `1e-3` in all three of the **x**, **y**, and **z** edit fields.

9 Click **OK**.

10 Click the **Go to Default 3D View** toolbar button to visualize the entire geometry of the snap hook and groove.

OPTIONS AND SETTINGS

Constants

1 From the **Options** menu select **Constants**.

2 Enter the constants from the following table.

NAME	EXPRESSION	DESCRIPTION
Displ_max	5e-3	Maximum displacement of the hook
E_hook	1e10	Young's modulus for the hook
nu_hook	0.35	Poisson's number for the hook
S_yield_hook	1.2e8	Yield stress for the hook
E_tan_hook	1.2e9	Isotropic tangent modulus for the hook

3 Click **OK**.

Integration Coupling Variables

To obtain the total force, necessary to insert and pull out the hook, you can integrate the variables lm1, lm2, and lm3 on the appropriate boundary. These are variables corresponding to the reaction forces in the x-, y-, and z-directions, respectively.

1 Select **Options>Integration Coupling Variables>Boundary Variables**.

2 Select Boundary 30 and enter the variables from the following table.

NAME	EXPRESSION
F_x	lm1
F_y	lm2
F_z	lm3

3 Click **OK**.

Expressions

1 From the **Options** menu select **Expressions>Scalar Expressions**.

2 In the **Name** edit field enter `Force` and in the **Expression** edit field enter `2*sqrt(F_x^2+F_y^2+F_z^2)`. The multiplication by 2 is due to symmetry.

3 Click **OK**.

PHYSICS SETTINGS

Application Mode Properties

Because of the large deformations present you need to turn on the large deformations options for the model. This ensures that the model uses the correct material model that can handle the large displacements. The weak constraints option adds the variables lm1, lm2, and lm3 for the reaction forces to the solution components. You can read more about reaction forces on page 96 of the *Structural Mechanics Module User's Guide*.

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

2 From the **Large deformation** list select **On**.

3 From the **Weak constraints** list select **On**.

4 Click **OK**.

Contact Pairs

The two solid bodies are not in contact initially and you need to specify which faces will be in contact by setting up a contact pair.

1 From the **Physics** menu select **Contact Pairs**.

2 In the **Contact Pairs** dialog box click the **New** button.

3 On the **Boundaries** page, in the **Master boundaries** list select the check boxes in front of Boundaries 1, 5, 6, 7, and 8.

4 In the **Slave boundaries** list select the check boxes in front of Boundaries 14, 15, and 20, 22, and 23.

5 Click **OK**.

Subdomain Settings

1 Select **Physics>Subdomain Settings** to open the **Subdomain Settings** dialog box.

2 Select Subdomains 2 and 3.

3 On the **Material** page select **Elasto-plastic** from the **Material model** list box.

4 In the **E** edit field enter `E_hook`.

5 In the ν edit field enter `nu_hook`.

6 Click the **Elasto-plastic material data** button.

7 In the σ_{ys} edit field enter `S_yield_hook`.

8 In the E_{Tiso} edit field enter `E_tan_hook`.

9 Click **OK**.

10 Click **OK**.

Boundary Settings

1 Select **Physics>Boundary Settings**.

2 From the **Boundary selection** list select Boundary 2.

3 On the **Constraint** page select **Fixed** from the **Constraint condition** list box.

4 Select Boundaries 3, 11, and 18.

5 On the **Constraint** page select **Symmetry plane** from the **Constraint condition** list box.

6 Select Boundary 30.

7 On the **Constraint** page select **Prescribed displacement** from the **Constraint condition** list box.

You can control the motion of the hook by the use of the solver parameter and boolean expressions in the expression for the x-displacement.

8 Select the R_x, R_y, and R_z check boxes.

9 In the R_x edit field enter `-Displ_max*((para<=1)*para+(para>1)*(2-para))`.

10 In the R_y and R_z edit fields enter `0`.

11 On the **Pairs** page select the contact pair Pair 1.

Since the hook and the groove are not in initial contact you need to specify the initial contact pressure to zero.

12 On the **Contact, Initial** page, enter `0` in the T_n edit field.

13 Click **OK**.

MESH GENERATION

You can mesh the slave boundaries in the contact pair finer than the master boundaries. By using the geometry scaling factors during meshing you can keep the number of elements down and thus reduce the size of the model.

1 Select **Mesh>Free Mesh Parameters**.

2 On the **Subdomain** page select Subdomains 2 and 3 from the **Subdomain selection** list.

3 Enter `1.1e-4` in the **Maximum element size** edit field.

4 Switch to the **Advanced** page.

5 Enter `0.5` in the **x-direction scale factor** edit field.

6 Enter `0.4` in the **z-direction scale factor** edit field.

7 Click **OK** to close the **Free Mesh Parameters** dialog box.

8 Click the **Mesh All (Free)** toolbar button.

COMPUTING THE SOLUTION

Solver Parameters

1 Click the **Solver Parameters** toolbar button.

2 On the **General** page, in the **Parameter name** edit field enter `para`.

3 In the **Parameter values** edit field enter `0 0.2:0.1:2`.

4 On the **Advanced** page, from the **Type of scaling** list box select **Manual**.

5 In the **Manual scaling** edit field enter `u 0.0005 v 0.0005 w 0.0005 lm1 1e7 lm2 1e7 lm3 1e7 Tn_cp1_smsld 5e7`.

Since the initial value of the contact pressure is zero, you need to use manual scaling of the variables. Read more about scaling of variables for contact problems on page 129 of the *Structural Mechanics Module User's Guide*.

6 Click **OK**.

Probe Plot

To monitor the force necessary to insert and retrieve the hook, you can create a probe plot. This plot is then displayed on the **Plot** page of the **Progress** dialog box, that COMSOL Multiphysics displays while the solver is running.

1 Select **Postprocessing>Probe Plot Parameters**.

2 In the **Probe Plot Parameters** dialog box click the **New** button.

3 In the dialog box that opens select **Global** from the **Plot type** list box.

4 In the **Plot name** edit field enter `Force plot`.

5 Click **OK**.

6 In the **Expression** edit field type `Force.`

7 Click **OK**.

8 Click the **Solve** toolbar button to solve the problem.

POSTPROCESSING AND VISUALIZATION

You can visualize the effective plastic strain in the hook by doing the following.

1 Click the **Plot Parameters** toolbar button.
2 Switch to the **Subdomain** page.
3 From the **Predefined quantities** list box select **Solid, Stress-Strain (smsld)>effective plastic strain**.
4 Click **OK** to close the dialog box and to update the plot.

Spherical Punch

Introduction

In this model a spherical punch is used to deform a circular plate, which is held in place by two jaws. Such a process is typically used for forming of sheet metal or for testing of material properties. The model demonstrates nonlinear structural analysis with contact, large deformations, and the use of elasto-plastic material model. This model is courtesy of CESI Ricerca, Dept. SSG, Milano, Italy.

Model Definition

By taking into account the axial symmetry of the problem you can build a 2D axisymmetric model to simulate the process.

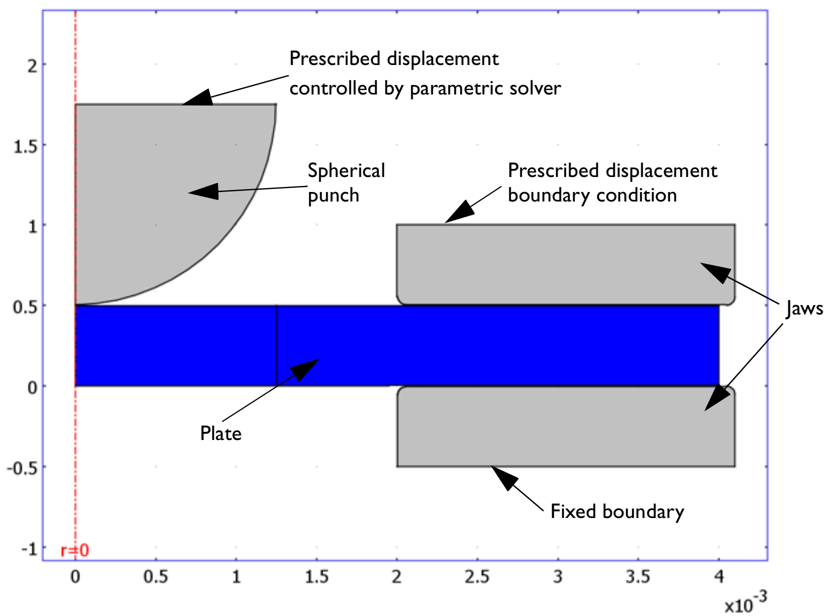


Figure 7-14: The geometry of the spherical punch and boundary conditions.

MATERIAL PROPERTIES

- Use the default material properties (steel) for the punch and the jaws.

- Use an elasto-plastic material model with kinematic hardening for the plate, with material properties outlined in the following table.

MATERIAL PARAMETER	VALUE
Young's modulus	$2\cdot10^5$ MPa
Poisson's number	0.33
Yield stress	300 MPa
Kinematic tangent modulus	$2\cdot10^4$ MPa

BOUNDARY CONDITIONS

- Apply a fixed boundary condition to the lower horizontal boundary of the jaw located below the plate.
- Apply a prescribed displacement boundary condition of –1 μm to the upper horizontal boundary of the jaw above the plate. This boundary condition models a force that maintains the upper jaw in contact with the plate. The easiest way to do this is to apply a displacement because a force probably causes convergence problems.
- Apply a prescribed displacement boundary condition to the upper boundary of the punch and use the solver parameter to gradually lower the sphere. The maximum displacement is –0.5 mm.

THE FINITE ELEMENT MESH

Create a mesh that takes into account the contact pairs in the model. For these, make sure that the slave boundaries are meshed at least twice as fine as the master boundaries. Since the main interest is the plastic deformation of the plate, mesh it with a finer mesh compared to the punch and the jaws.

Results

Figure 7-15 shows the von Mises stress distribution in the plate at the final parameter step, which corresponds to 0.5 mm displacement. Note that the highest stresses are on the bottom surface of the plate, below the punch. This is where the material reaches the yield stress first.

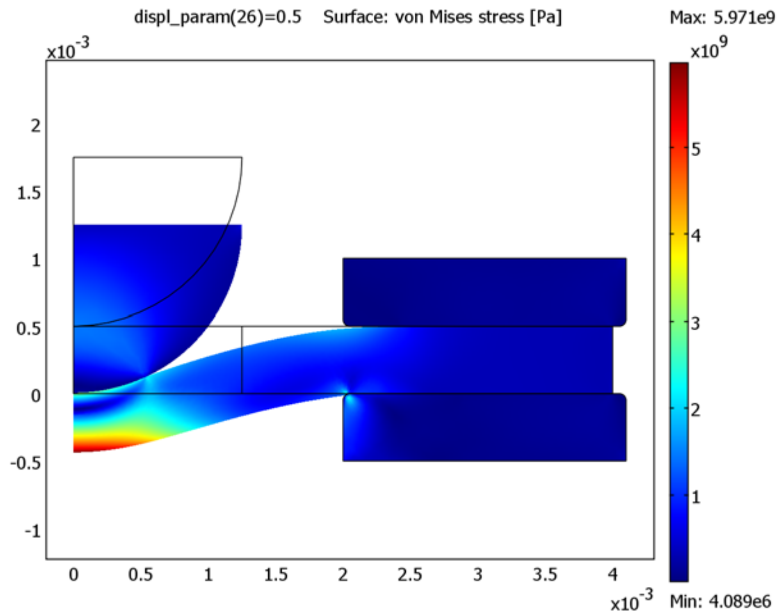


Figure 7-15: The von Mises stress distribution at the maximum displacement of the punch.

Figure 7-16 displays the force necessary to deform the plate plotted against the displacement parameter. The force-displacement relationship is linear initially, until the material in the plate reaches the yield point.

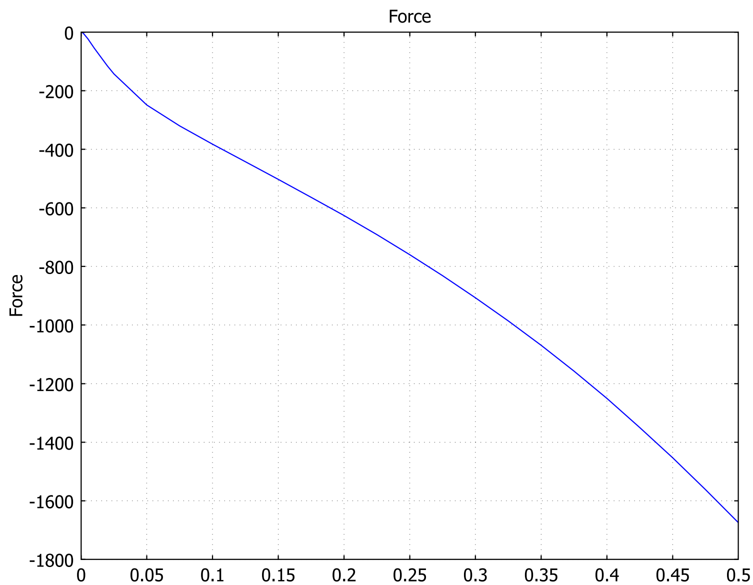


Figure 7-16: The force on the punch versus the displacement parameter.

Model Library path: `Structural_Mechanics_Module/Contact_and_Friction/spherical_punch`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **Axial symmetry (2D)** from the **Space dimension** list.

2 From the list of application modes select **Structural Mechanics Module> Axial Symmetry, Stress-Strain>Parametric Analysis**.

3 Click **OK**.

IMPORT OF GEOMETRY

1 Choose **File>Import>CAD Data From File**.
2 In the **Look in** list browse to the folder `models/Structural_Mechanics_Module/Contact_and_Friction` in your COMSOL Multiphysics installation directory.
3 Select the file `spherical_punch.mphbin`, then click **Import**.

   Before leaving the draw mode you need to set the assembly mode in order to be able to define the contact pairs between the different objects.
4 From the **Draw** menu select **Use Assembly**.

OPTIONS AND SETTINGS

Constants

1 From the **Options** menu select **Constants**.
2 Enter the constants from the following table:

NAME	EXPRESSION	DESCRIPTION
`Displ_scale`	`-1[mm]`	`Displacement scale`
`displ_param`	`0`	`Displacement multiplier`
`E_plate`	`2e5[MPa]`	`Young's modulus for the plate`
`nu_plate`	`0.33`	`Poisson's ratio for the plate`
`S_yield_plate`	`300[MPa]`	`Yield stress for the plate`
`E_tan_plate`	`2e4[MPa]`	`Kinematic tangential modulus for the plate`

3 Click **OK**.

Integration Coupling Variables

To obtain the force which is necessary to push down the punch you can integrate the variable lm2 which corresponds to the reaction force in this direction. This variable is automatically added to the solution components when you turn on weak constraints for the model. The variable you define here is available for postprocessing or to use in any expression. You can read more about reaction forces on page 96 of the *Structural Mechanics Module User's Guide*.

1 Select **Options>Integration Coupling Variables>Boundary Variables**.
2 Select Boundary 14.
3 In the **Name** column enter `Force`.

4 In the **Expression** column enter `2*pi*lm2`.

The factor 2π helps to take into account the integration around the circumferential direction for this 2D axisymmetric model.

5 Click **OK**.

PHYSICS SETTINGS

Contact Pairs

1 Select **Physics>Contact Pairs** to open the **Contact Pairs** dialog box.

2 Click **New** to create a new contact pair.

3 In the **Master boundaries** list select the check box in front of Boundaries 17, 19, and 21.

4 In the **Slave boundaries** list select the check box in front of Boundaries 9 and 10.

5 By repeating Steps 2–3, create two more contact pairs according to the table below.

CONTACT PAIR	MASTER BOUNDARIES	SLAVE BOUNDARIES
Pair 2	25, 27	8
Pair 3	15, 16	3, 5

6 Click **OK**.

Application Mode Properties

By turning on large deformations for the model you make sure that the correct formulation of the underlying equations that can handle large displacements are used to solve the problem. The weak constraints option in the model adds variables for the reaction forces to the solution components, which you can use to compute the force needed to push down on the punch.

1 Select **Physics>Properties**.

2 From the **Large deformation** list box select **On**.

3 From the **Weak constraints** list box select **On**.

4 Click **OK**.

Subdomain Settings

1 Select **Physics>Subdomain Settings**.

2 From the **Subdomain selection** list select Subdomains 1 and 2.

3 Enter `E_plate` in the **E** edit field.

4 Enter `nu_plate` in the ν edit field.

5 From the **Material model** list select **Elasto-plastic**.

6 Click the **Elasto-plastic material data** button.

7 From the **Hardening model** list select **Kinematic**.

8 Enter `S_yield_plate` in the σ_{ys} edit field.

9 Enter `E_tan_plate` in the $\mathbf{E_{Tkin}}$ edit field.

10 Click **OK**.

11 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Settings

1 Select **Physics>Boundary Settings**.

2 From the **Boundary selection** list select Boundary 14.

3 From the **Constraint condition** list box select **Prescribed displacement**.

4 Select the $\mathbf{R_r}$ and $\mathbf{R_z}$ check boxes.

5 Enter `Displ_scale*displ_param` in the $\mathbf{R_z}$ edit field.

6 From the **Boundary selection** list select Boundary 18.

7 From the **Constraint condition** list box select **Fixed**.

8 From the **Boundary selection** list select Boundary 24.

9 From the **Constraint condition** list box select **Prescribed displacement**.

10 Select the $\mathbf{R_r}$ and $\mathbf{R_z}$ check boxes.

11 Enter `-1e-6` in the $\mathbf{R_z}$ edit field.

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

MESH GENERATION

1 Select **Mesh>Free Mesh Parameters**.

2 On the **Subdomain** page select Subdomains 1 and 2 from the **Subdomain selection** list.

3 Enter `8e-5` in the **Maximum element size** edit field.

4 On the **Boundary** page select boundaries and enter settings on the **Parameters** page according to the table below.

BOUNDARIES	MAXIMUM ELEMENT SIZE
15, 16	`5e-5`
1, 5, 7, 8, 9, 10	`2e-5`
3	`1e-5`

5 From the **Boundary selection** list select Boundaries 19 and 25.

6 On the **Distribution** page select the **Constrained edge element distribution** check box.

7 Select the **Edge vertex distribution** radio button and enter `0:1/38:1` in the edit field.

8 From the **Boundary selection** list select Boundaries 21, 22, 27, and 28.

9 On the **Distribution** page select the **Constrained edge element distribution** check box.

10 Click the **Number of edge elements** button and enter `2` in the edit field.

11 Click **OK** to apply the settings and close the **Free Mesh Parameters** dialog box.

12 Click the **Mesh All (Free)** button on the Mesh toolbar.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** toolbar button.

2 On the **General** page enter `displ_param` in the **Parameter name** edit field.

3 Enter `0 0.001 0.002 0.005 0.01 0.02 0.025:0.025:0.5` in the **Parameter values** edit field.

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

5 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 Select **Postprocessing>Global Variables Plot**.

2 Enter `Force` in the **Expression** edit field.

3 Click the **Add Entered Expression** button.

4 Click **OK** to close the dialog box and to plot the force versus the displacement parameter.

8

Dynamics and Vibration Models

This chapter presents models that simulate dynamics and vibration.

Rotor

This model contains an eigenfrequency analysis of a rotor in an electric motor. In the design of a motor it is important that no eigenfrequencies for the rotor lie within the operating interval of the revolution speed (in revolutions per second) for the motor.

Model Definition

GEOMETRY

The geometry is fully axisymmetric, so you can create a cross section about the symmetry axis and rotate it to create the full 3D geometry.

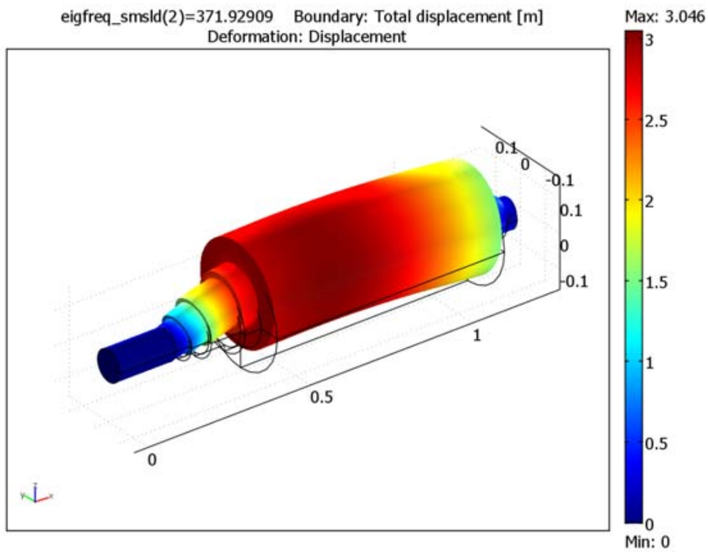
MATERIAL PROPERTIES AND BOUNDARY CONDITIONS

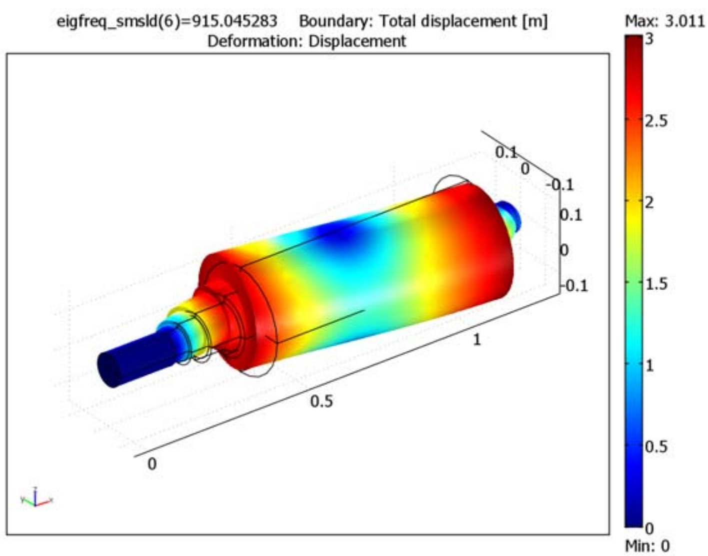
The rotor is made of solid steel. The only settings that you need to change are the boundary conditions:

- The left end of the rotor is fully fixed.
- The left end is constrained to only rotate about and slide along the axis of the rotor. This means that the boundary condition is zero displacement in the normal direction.

Results

The analysis provides the six first eigenfrequencies, of which the 2nd and 3rd are the same. The plots below show the 2nd and 6th eigenmodes. They correspond to eigenfrequencies of 366 and 905 Hz.





Model Library path: `Structural_Mechanics_Module/Dynamics_and_Vibration/rotor`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Click the **New** button or start COMSOL Multiphysics to open the **Model Navigator**.
2. Select **3D** in the **Space dimension** list.
3. In the list of application modes choose **Structural Mechanics Module>Solid, Stress-Strain>Eigenfrequency analysis**.
4. Click **OK**.

GEOMETRY MODELING

Create the rotor by revolving a cross section about the symmetry axis.

1. From the **Draw** menu, choose **Work-Plane Settings**.
2. In the **Work-Plane Settings**, click **OK** to create a work plane in the x-y plane (the default).
3. From the **Options** menu, choose **Axes/Grid Settings**.
4. Set axis and grid settings in the work plane geometry (Geom2) according to the following table. To specify the grid settings, click the **Grid** tab and clear the **Auto** check box.

AXIS		GRID	
x min	`-0.2`	x spacing	`0.2`
x max	`1.4`	Extra x	`0.171 0.213 0.266 0.356 0.406 1.106 1.186 1.226`
y min	`-0.4`	y spacing	`0.1`
y max	`0.4`	Extra y	`-0.1475 -0.083 -0.066 -0.055 -0.053 -0.045 -0.0425`

5. Click **OK**.
6. Shift-click the **Line** button on the Draw toolbar to open the **Line** dialog box.
7. In the **x** edit field type `0 0 0.171 0.213 0.213 0.266 0.266 0.356 0.356 0.406 0.406 1.106 1.106 1.186 1.186 1.226 1.226` and in the **y** edit field

type `0 -0.045 -0.045 -0.045 -0.055 -0.055 -0.066 -0.066 -0.083 -0.083 -0.1475 -0.1475 -0.053 -0.053 -0.0425 -0.0425 0`. From the **Style** list select **Closed polyline (solid)**, then click **OK**.

8 From the **Draw** menu, choose **Revolve**.

9 Use the x-axis as the revolution axis. Specify this by typing `1` and `0` in the **x** and **y** edit fields under the **Second point** button to define the second point on the revolution axis.

10 Click **OK** to revolve the cross section 360 degrees about the x-axis.

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

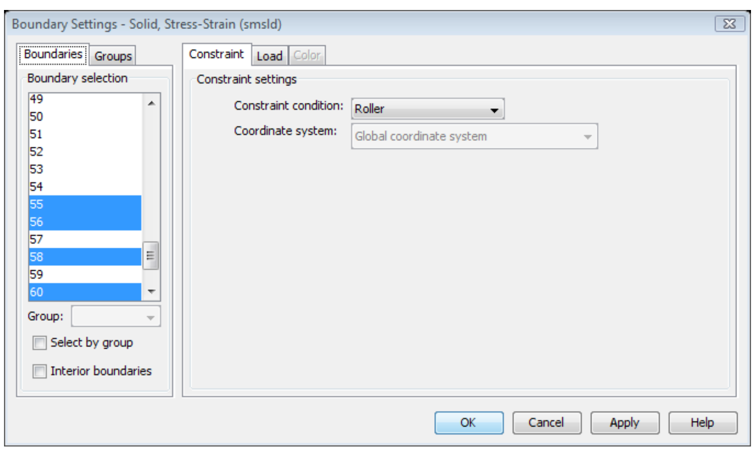
PHYSICS SETTINGS

Boundary Conditions

1 From the **Physics** menu choose **Boundary Settings**.

2 On the **Constraint** page of the **Boundary Settings** dialog box specify constraints according to the following table.

BOUNDARY	2, 4, 6, 8	55, 56, 58, 60
Constraint condition	Fixed	Roller



Subdomain Settings

The rotor is made of steel so you can use the default settings for the material properties.

MESH GENERATION

Click the **Initialize Mesh** button to create a mesh using the default mesh settings.

COMPUTING THE SOLUTION

Solve the problem by clicking the **Solve** toolbar button. With the default settings, COMSOL Multiphysics computes the first six eigenmodes and eigenfrequencies.

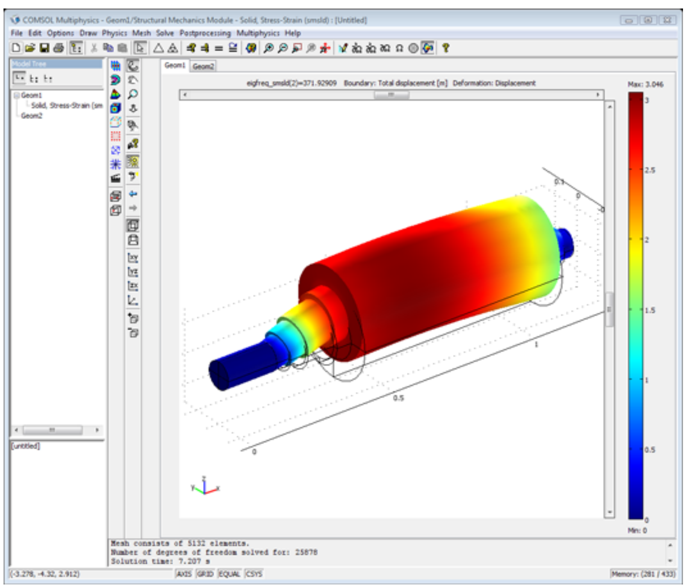
POSTPROCESSING AND VISUALIZATION

COMSOL Multiphysics plots the first eigenmode using a slice plot. To change the plot to show the deformed shape and other eigenmodes, follow these steps:

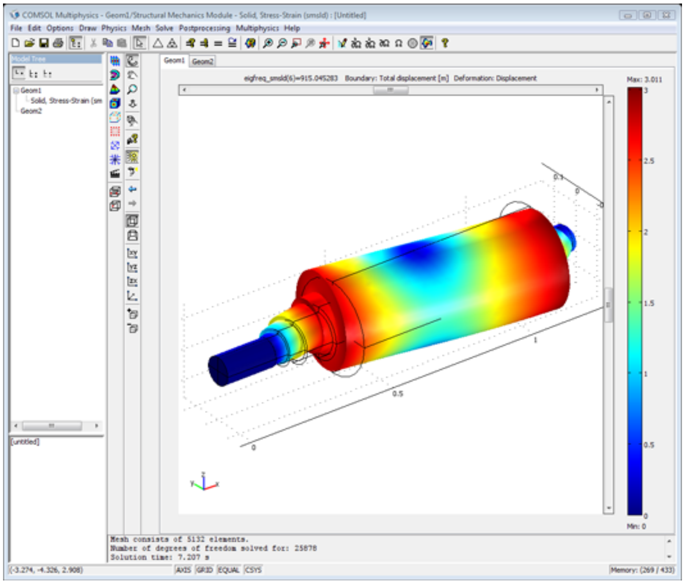
1 **Postprocessing** menu, choose **Plot Parameters**.

2 In the **Plot Parameters** dialog box, clear the **Slice** check box and select the **Boundary** and **Deformed shape** check boxes in the **Plot type** area.

3 Select the second eigenfrequency in the **Eigenfrequency** list and click **Apply** to plot the second eigenmode.



4 Select the last eigenfrequency in the **Eigenfrequency** list and click **OK** to plot the sixth eigenmode.



Eigenfrequency Analysis of a Free Cylinder

Introduction

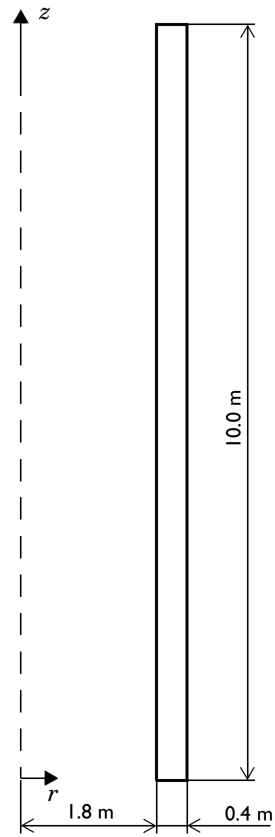
In the following example you will build and solve an axial symmetric model using the Axial Symmetry, Stress-Strain application mode.

This model calculates the eigenfrequencies and mode shapes of an axisymmetric free cylinder. The model is taken from NAFEMS Free Vibration Benchmarks (Ref. 1). The eigenfrequencies are compared with the values given in the benchmark report.

Model Definition

The model is NAFEMS Test No 41, "Free Cylinder" described on page 41 in NAFEMS *Free Vibration Benchmarks, Volume 3* (Ref. 1). The Benchmark tests the capability to handle rigid body modes and close eigenfrequencies.

The cylinder is 10 m high with an inner radius of 1.8 m and a thickness of 0.4 m.



MATERIAL

Isotropic material with E=2.0·10^{11} Pa and ν=0.3.

LOADS

In an eigenfrequency analysis loads are not needed.

CONSTRAINTS

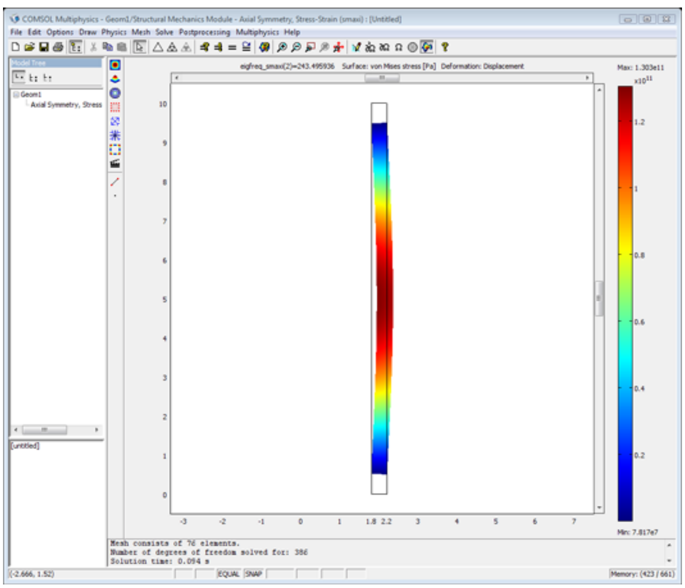
No constraints are applied because the cylinder is free.

Results

The rigid body mode with an eigenvalue close to zero is found. The eigenfrequencies are in close agreement with the target values from the NAFEMS Free Vibration Benchmarks (Ref. 1).

EIGENFREQUENCY	COMSOL	TARGET [1]
f_1	0 Hz	0 Hz
f_2	243.50 Hz	243.53 Hz
f_3	377.40 Hz	377.41 Hz
f_4	394.24 Hz	394.11 Hz
f_5	397.88 Hz	397.72 Hz
f_6	405.46 Hz	405.28 Hz

The following plot shows the shape of the second eigenmode:



Reference

1. Abassian, F., Dawswell, D. J., and Knowles, N. C., *Free Vibration Benchmarks, Volume 3*, NAFEMS, Glasgow, 1987.

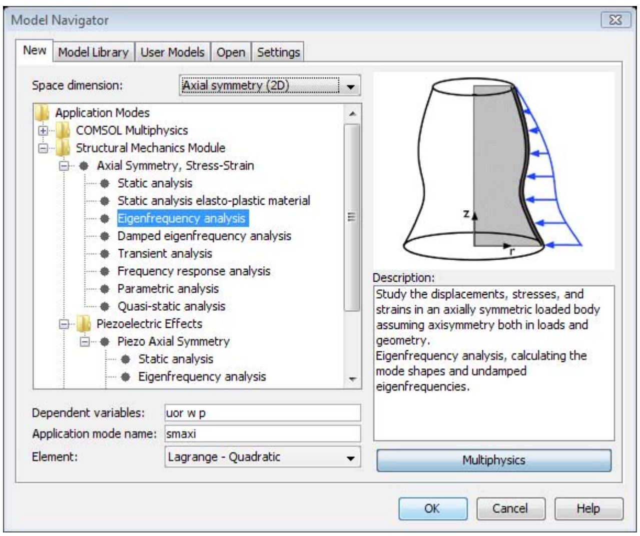
Model Library path: `Structural_Mechanics_Module/Dynamics_and_Vibration/free_cylinder`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **Axial symmetry (2D)** in the **Space dimension** list on the **New** page in the **Model Navigator.**

2 Select **Structural Mechanics Module>Axial Symmetry, Stress-Strain>Eigenfrequency analysis** and click **OK**.



OPTIONS AND SETTINGS

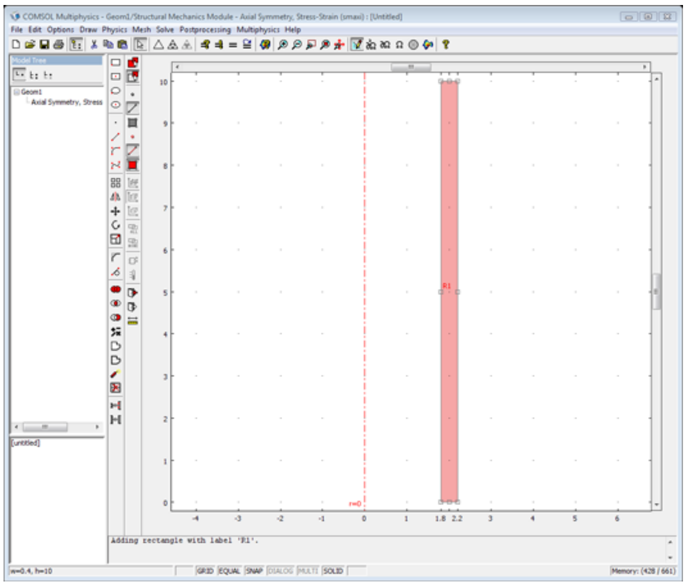
1 Select **Axes/Grid Settings** from the **Options** menu and give axis and grid settings according to the following table:

AXIS		GRID	
`r min`	`-0.2`	`r spacing`	`1`
`r max`	`2.4`	`Extra r`	`1.8 2.2`

AXIS		GRID	
`z min`	`-0.2`	`z spacing`	`1`
`z max`	`10.2`	`Extra z`	

GEOMETRY MODELING

1 Click the **Rectangle/Square** toolbar button and click the left mouse button at (1.8, 0) move the mouse to (2.2, 10.0) and click the left mouse button again to create the rectangle.



PHYSICS SETTINGS

Boundary Settings

No boundary conditions need to specified as the cylinder is free and without loads.

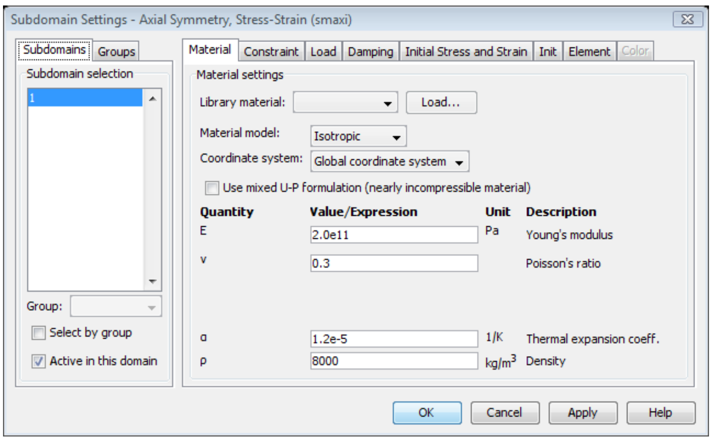
Subdomain Settings

Specify the material properties of the free cylinder.

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

2 Specify subdomain settings according to the following table:

SUBDOMAIN	1	
Page	Material	
	Material model	Isotropic material
	E	2.0E11 (Pa)
	ν	0.3
	ρ	8000 (kg/m^3)



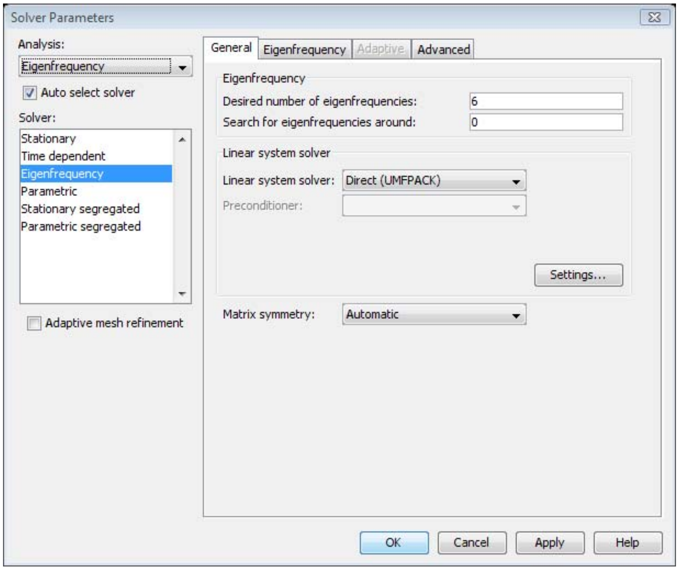
MESH GENERATION

Use the default mesh.

COMPUTING THE SOLUTION

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

2 The eigenfrequency solver is already selected because you chose an eigenfrequency analysis in the **Model Navigator.** Regarding the eigenfrequency solver parameters, the default value of searching for the first 6 eigenfrequencies is OK.



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

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

POSTPROCESSING AND VISUALIZATION

Look at the eigenfrequencies and mode shapes.

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

2 On the **General** page, look in the **Eigenfrequency** list to find the eigenfrequencies.

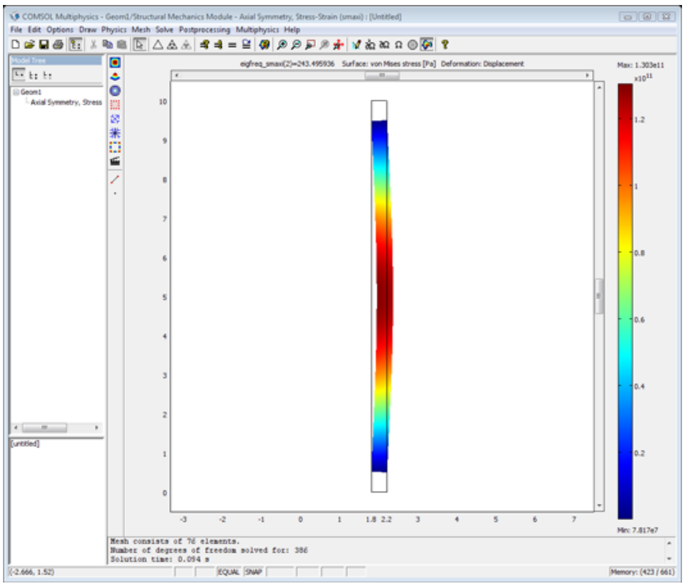
EIGENFREQUENCY	COMSOL
f_1	0 Hz
f_2	243.50 Hz
f_3	377.40 Hz
f_4	394.24 Hz
f_5	397.88 Hz
f_6	405.46 Hz

In an axisymmetric model that is free to move there is one rigid body mode. The rigid body mode has an eigenfrequency close to zero. The rigid body mode shape is a pure axial rigid body translation without any radial displacement.

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

4 Select the **Deformed shape** check box on the **General** page to plot the second or first true mode shape.

5 Click **OK** to plot the first true eigenmode.



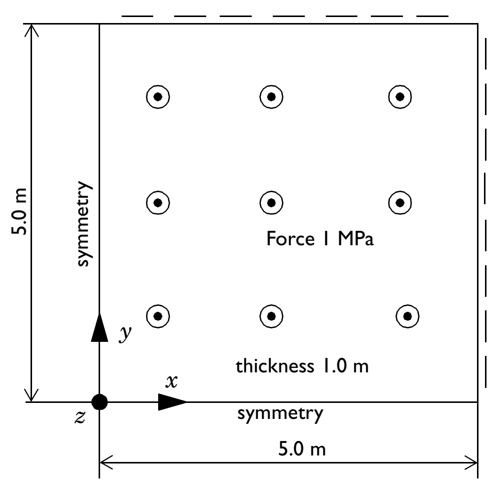
Frequency Response Analysis of a Simply Supported Plate

In the following example you will build and solve a solid mechanics model using the Mindlin Plate application mode.

The model is a simply-supported thick square plate 10 m wide exposed to a uniform harmonic pressure. The model is taken from NAFEMS Test no 21 H, "Simply-supported thick square plate harmonic forced vibration response" described on page 29 in *NAFEMS Selected Benchmarks for Forced Vibration* (Ref. 1). The computed maximum displacement, stress, and excitation frequency that results in maximum response are compared with the values given in the benchmark report.

Model Definition

The geometry is a square. Due to symmetry in load and geometry, only a quarter of the square is analyzed.



MATERIAL

- Isotropic material with E = 2.0E11 Pa, ν = 0.3, and ρ = 8000 kg/m^3
- Rayleigh damping: α_{dM} = 5.772, β_{dK} = $6.929 \cdot 10^{-5}$

LOADS

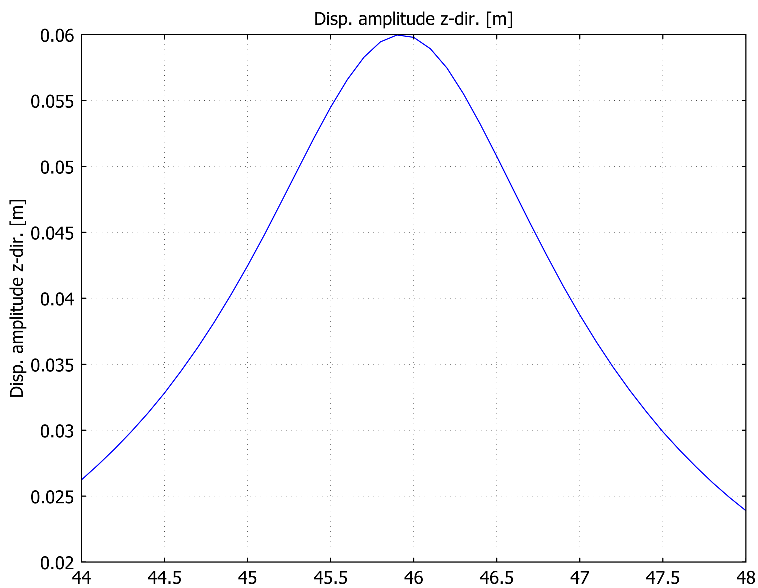
A distributed harmonic force of 1 MPa on the upper surface pointing in the positive z direction. The excitation frequency range is 44–48 Hz.

CONSTRAINTS

Simply-supported edges on the top and left edge, global rotations constrained on the symmetry edges, $x = 0$, $y = 0$.

Results

The displacement amplitude at the center as a function of the excitation frequency is



The following table shows a comparison between the solution and the NAFEMS benchmark (Ref. 1):

RESULT	COMSOL MULTIPHYSICS	NAFEMS (REF. 1)
Maximum displacement (mm)	59.66	58.33
Stress at top of plate at center (MPa)	802.8	800.8
Frequency at maximum response (Hz)	45.9	45.90

All results are in close agreement with the benchmark.

Reference

1. J. Maguire, D.J. Dawswell, and L. Gould, *Selected Benchmarks for Forced Vibration*, NAFEMS, Glasgow, 1989.

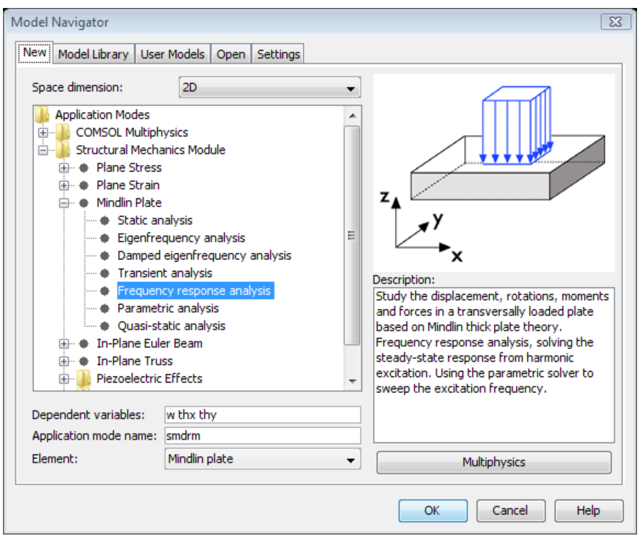
Model Library path: `Structural_Mechanics_Module/Dynamics_and_Vibration/harmonically_excited_plate`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **2D** in the **Space dimension** list on the **New** page in the **Model Navigator**.

2 Select **Structural Mechanics Module>Mindlin Plate>Frequency response analysis** and click **OK**.



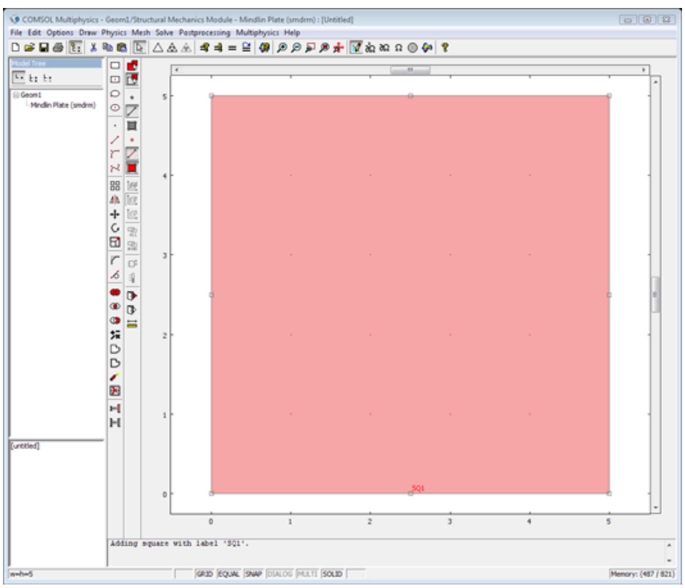
GEOMETRY MODELING

1 Select **Axes/Grid Settings** from the **Options** menu and give axis and grid settings according to the following table:

AXIS		GRID	
x min	-5	x spacing	1
x max	5	Extra x	
y min	-5	y spacing	1
y max	5	Extra y	

2 Click the **Rectangle/Square** toolbar button and click the right mouse button at (0, 0). Move the mouse while holding down the right mouse button, release the mouse button at (5, 5) to create a square.

3 Click the **Zoom Extents** button on the Main toolbar to zoom in on the created square.



PHYSICS SETTINGS

In this section the analysis type is specified, and the boundary and subdomain settings is made.

Boundary Settings

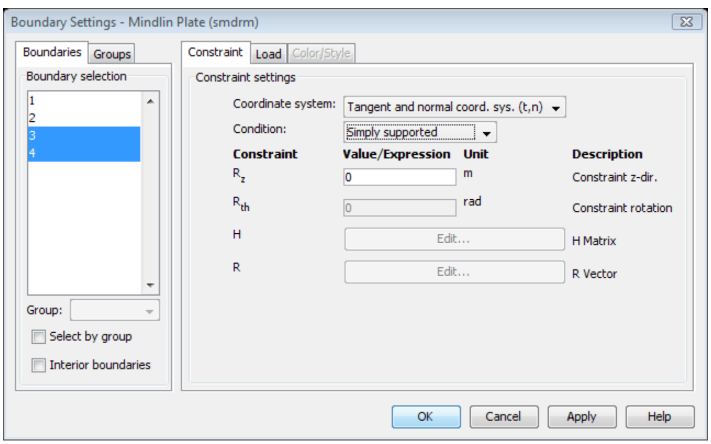
Constrain the outer edges as simply supported. Specify the symmetry edges to have the global rotation corresponding with the symmetry constrained.

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

2 Specify boundary settings according to the following tables:

BOUNDARY	3. 4	
Page	Constraint	
	Coordinate system	Tangent and normal coord. sys. (t,n)
	Condition	Simply supported

BOUNDARY	1		2	
Page	Constraint		Constraint	
	Coordinate system	Global coordinate system	Coordinate system	Global coordinate system
	R_y	0	R_x	0



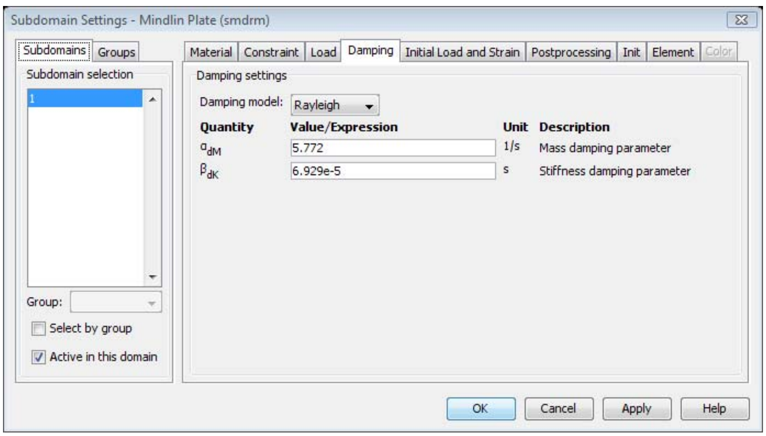
Subdomain Settings

Specify the material properties, damping properties, and load of the Mindlin plate.

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

2 Specify subdomain settings according to the following table:

SUBDOMAIN	1		1		1	
Page	Material		Load		Damping	
	Material model	Isotropic material	F_z	1e6	Damping model	Rayleigh
	E	2.0E11			α_{dM}	5.772
	ν	0.3			β_{dK}	6.929E-5
	ρ	8000				
	thickness	1				



MESH GENERATION

Use the default mesh settings. Click the **Initialize Mesh** button on the Main toolbar to create a mesh.

COMPUTING THE SOLUTION

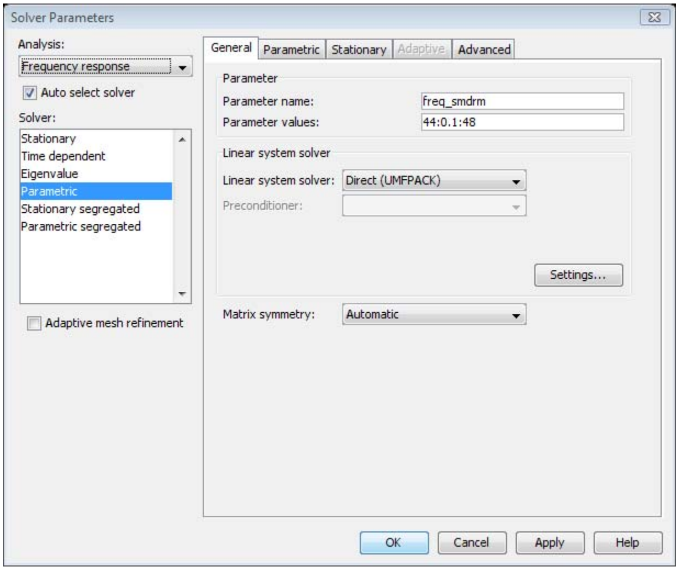
Specify the excitation frequency and range:

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

The parametric solver is the solver connected to the frequency response analysis type, and the **Auto select solver** option make sure that it has already been selected.

2 Type `freq_smdrm` in the **Parameter name** edit field.

3 Type `44:0.1:48` in the **Parameter values** to perform a frequency sweep between 44 Hz and 48 Hz in 0.1 Hz increments.



4 Click **OK**.

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

POSTPROCESSING AND VISUALIZATION

Analyze the maximum displacement amplitude.

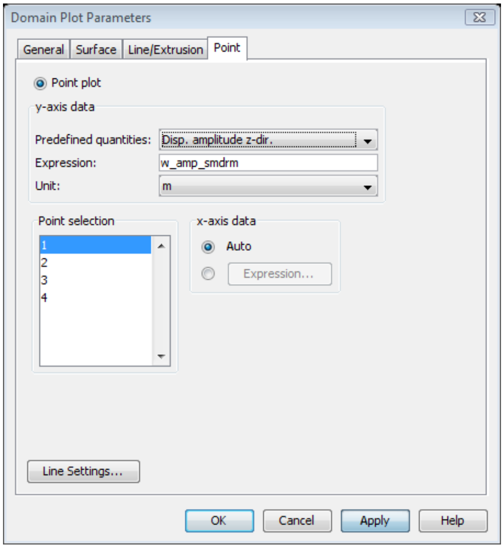
1 Select **Domain Plot Parameters** from the **Postprocessing** menu.

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

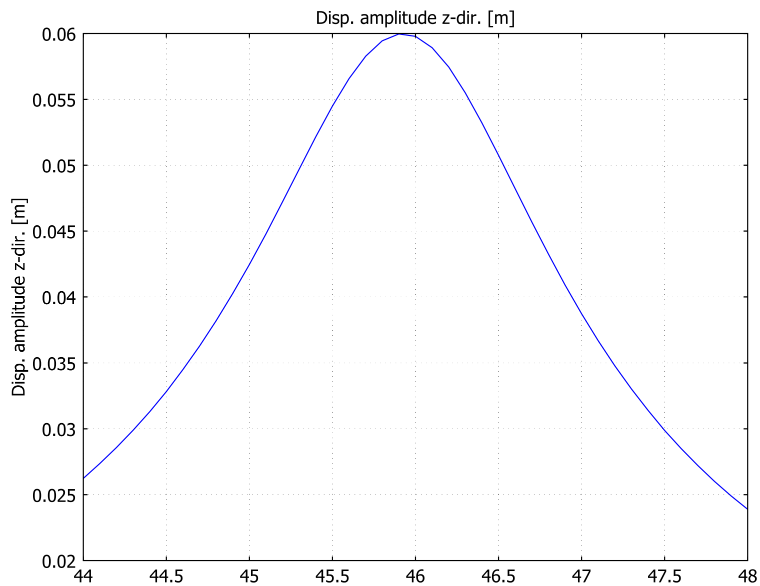
3 Select the **Point** page.

4 Select **Disp. amplitude z dir.** from the **Predefined quantities** list.

5 Select 1 from the **Point selection** list to look at the result in the middle of the plate or select the point by clicking on it in the GUI.



6 Click **Apply** to perform the selected plot, click **Cancel** to close the dialog box.



The maximum displacement amplitude is about 60 mm and appears at 45.9 Hz.

Find the maximum stress at the center of the plate:

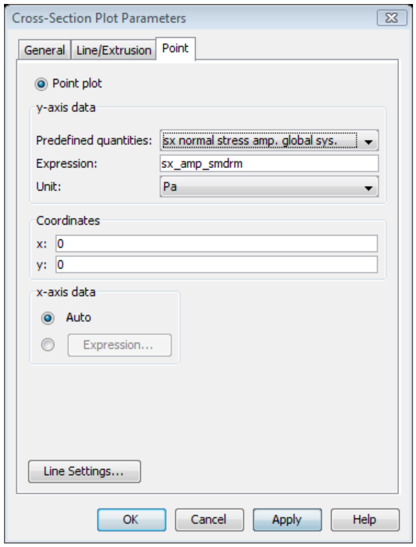
1 Select **Cross-Section Plot Parameters** from the **Postprocessing** menu.

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

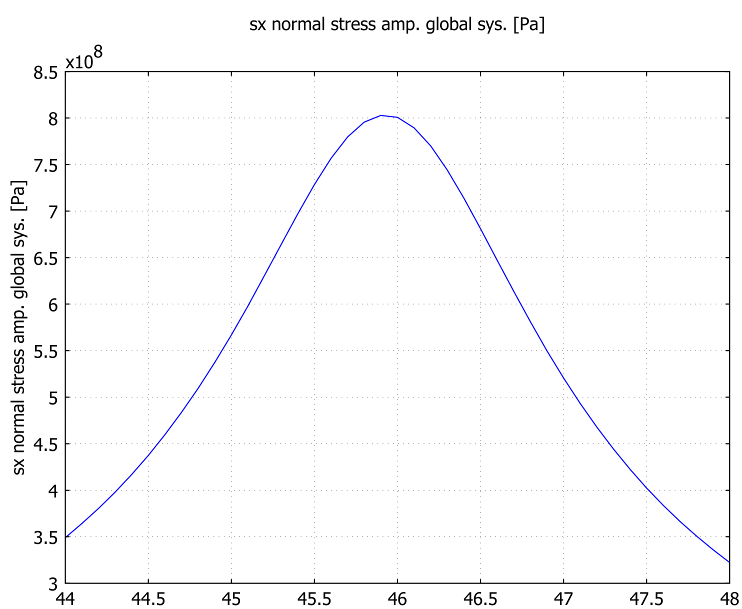
3 Select the **Point** page.

4 Select **sx normal stress amp. global sys.** from the **Predefined quantities** list.

5 The default **x** and **y** coordinates coincide with the request to look at the result in the middle, x=0, y=0.



6 Click **Apply** to perform the selected plot, click **Cancel** to close the dialog box.



The maximum stress is just above 800 MPa.

Look at the maximum displacement in a surface plot.

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

2 Go to the **General** page and select **45.9** from the **Parameter value** list in the **Solution to use** area.

3 Click the **Surface** tab.

4 Select **Disp. amplitude z dir.** from the **Predefined quantities** list on the **Surface Data** tab.

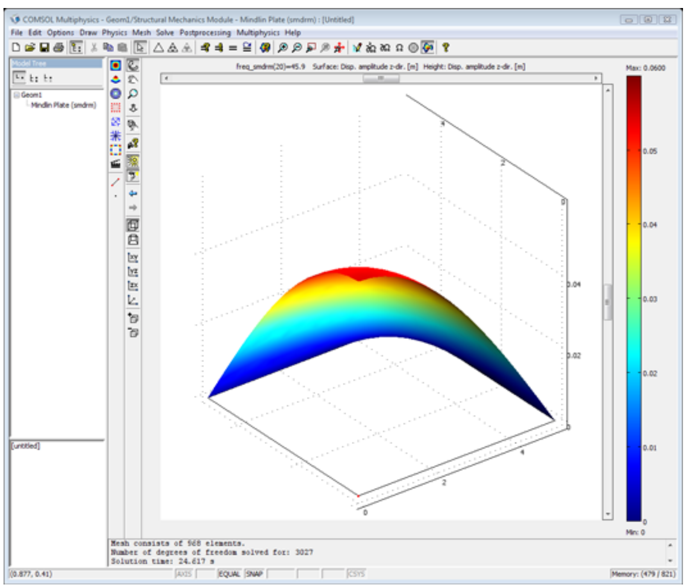
5 Click the **Height Data** tab and select the **Height data** check box.

6 Select **Disp. amplitude z dir.** from the **Predefined quantities** list on the **Height Data** tab.

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

8 Click the **Scene Light** and **Headlight** button on the Camera toolbar.

9 Click the **Zoom Extents** button on the Main toolbar



9

Fatigue Models

This chapter contains a number of models using fatigue analysis. Note that the commands for fatigue analysis require COMSOL Script or MATLAB. The table below lists the models categorized by the type of fatigue analysis performed.

MODEL	PAGE	FATIGUE SCRIPT	HIGH CYCLE FATIGUE	LOW CYCLE FATIGUE	PROP. LOADING	NONPROP. LOADING
Shaft with Fillet	346	`shaft_with_fillet_fatigue.m`	√			√
Frame with Cutout	357	`frame_with_cutout_fatigue.m`	√		√	
Cylinder with Hole	372	`cylinder_hole_linear_fatigue.m` `cylinder_hole_plastic_fatigue.m`		√		√
Fatigue Analysis of Automobile Wheel Rim	386	`wheel_rim_fatigue_superpose.m` `wheel_rim_fatigue_analysis.m`	√			√

Shaft with Fillet

Introduction

This model is of "benchmark" type, based on the example found in section 5.4.3 of Ref. 1, and shows how to perform a high-cycle fatigue analysis for nonproportional loading using critical planes.

Model Definition

The geometry is a circular shaft with two different diameters, 10 mm and 20 mm. At the transition, between the two parts, there is a fillet with a radius of 3 mm.

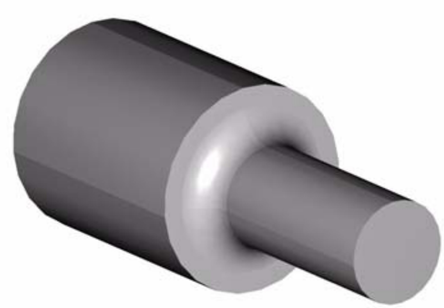


Figure 9-1: Model geometry

Two time-dependent loads are applied to the small end of the shaft: a transverse force, causing bending, and a twisting moment. As Figure 9-2 shows, the force varies between 0 and 2.95 kN and the torque between –30.3 Nm and +30.3 Nm.

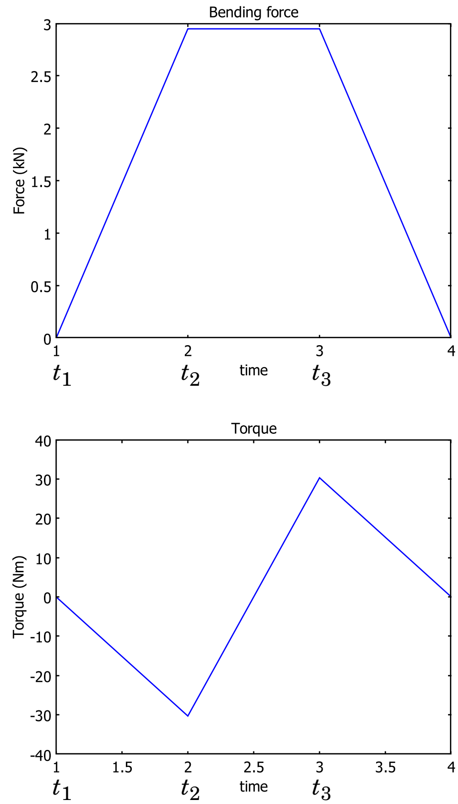


Figure 9-2: Variation of the bending force and twisting moment during one load cycle.

Compute the fatigue usage factor by analyzing the total stress distribution at times t_1, t_2 and t_3.

MATERIAL PROPERTIES

- Elastic data: Isotropic with E = 100 GPa, ν = 0
- Fatigue data: The fatigue limit is known for two cases with pure axial loading. For pure tension it is 560 MPa (σ_{max} = 1120 MPa, σ_{min} = 0 MPa), and for fully reversed loading it is 700 MPa (σ_{max} = 700 MPa, σ_{min} = –700 MPa). This gives

the Findley parameters f = 440 MPa and k = 0.23 if Equation 14-16 on page 379 in the *Structural Mechanics Module User's Guide* is applied.

CONSTRAINTS

The thick end of the bar is fixed.

LOADS

- Bending load: The transverse force is evenly distributed as a shear traction over the small end.
- Torsional load: A shear stress in the circumferential direction is applied to the bar end. It is given as a linear variation with the radius in correspondence with analytical solutions for torsion.

Results and Discussion

Figure 9-3 shows the von Mises stresses for the bending load cases. The location for the maximum effective stress is at the surface of the fillet, at a radius slightly larger than the minimum radius of the shaft. The maximum effective stress can also be found at a diametrically opposite location. Here, the bending load gives rise to a compression stress of equal magnitude as the tension stress at the other side. Due to numerical reasons, the maximum stresses do not occur exactly in the x-z plane.

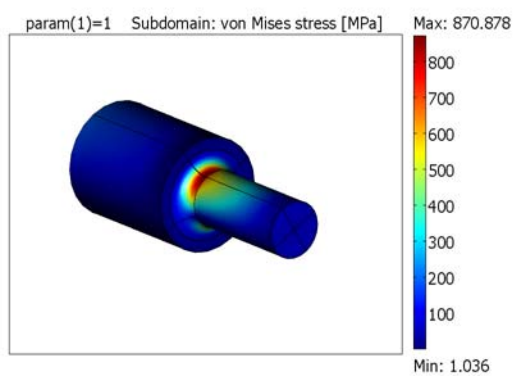


Figure 9-3: von Mises stress distribution from the bending load case.

Figure 9-4 shows the effective stress distribution for the torsional load. You can find the maximum value along the surface of the fillet, where the shear stress in the circumferential direction is at a maximum.

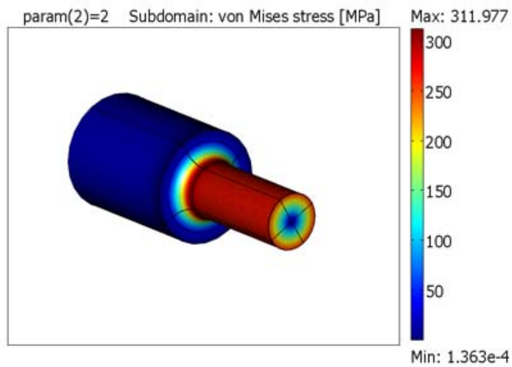


Figure 9-4: von Mises stress distribution for the torsional load case.

In Figure 9-5 you can see that the highest value of the utilization factor, 0.962, is found where the positive bending stress is combined with the shear stress from torsion. Note that on the diametrically opposite side of the shaft, with the maximum compressive stress due to the bending, the utilization factor is only slightly increased compared to the surrounding areas.

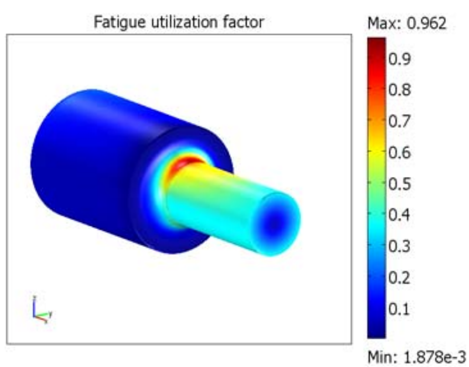


Figure 9-5: Fatigue utilization factor.

The history of each of the global stress components in the most critical point is shown in Figure 9-6. Because the peak stresses occur at a small distance up along the fillet and because, for numerical reasons, they are not found exactly in the plane $y = 0$, all stress components are nonzero.

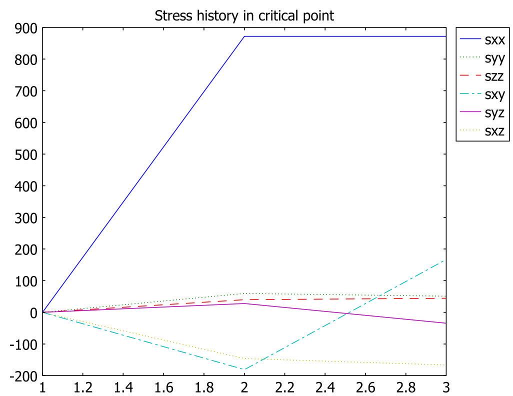


Figure 9-6: Histories for global stress components.

For the example in Ref. 1 the computed Findley parameter is 433 MPa, which gives a utilization factor of 0.984. This is slightly higher than the utilization factor computed by this model, which is 0.962. The difference is mainly due to the simplified approach of the referenced example, where only the normal stress component from the bending load and the shear stress component from the torsional load are considered. In reality, there are additional stress components, especially in the bending case. Also, the peak stress concentration does not occur at exactly the same location in the two load cases.

Modeling in COMSOL Multiphysics

Start by using the parametric solver to solve the static problem for the load cases of the maximum bending force and the maximum twisting moment, respectively. These are the two basic load cases that you can combine and use for the fatigue analysis.

Carry out the fatigue analysis, in either COMSOL Script or MATLAB, by doing the following steps:

- Extract the stress distribution (stress tensor) for both basic load cases from the FEM structure of the model.

- Set up the matrix containing the combinations of basic load cases which you can use for the fatigue analysis. The rows of this matrix correspond to the time instances you are analyzing, and the columns correspond to the basic load cases.

  In this case the matrix is a 3x2 matrix. If you look at Figure 9-2 you can see that at time t_1 both loads are zero, thus the first row has only zeros. At time t_2 the bending force is at maximum, while the torque at minimum. You obtain the combined stresses by subtracting the solution of the twisting moment from that of the bending force. The second row of the matrix is thus $1\ -1$. Both loads are at maximum at the third time, t_3, which means that you can add the basic load cases and the third row of the matrix contains two ones.
- Run the fatigue analysis function `hcfmultiax`.

Reference

1. D, F. Socie and G.B. Marquis, *Multiaxial Fatigue*, SAE, 1999.

Model Library path: `Structural_Mechanics_Module/Fatigue/shaft_with_fillet`

Note: This model requires COMSOL Script or MATLAB to run the fatigue script `shaft_with_fillet_fatigue.m`

Modeling Using the Graphical User Interface

This section describes how to solve the two basic load cases using COMSOL Multiphysics.

MODEL NAVIGATOR

1. On the **New** page, select **3D** from the **Space dimension** list.
2. From the **Application Modes** list, select **Structural Mechanics Module>Solid, Stress-Strain>Static analysis**.
3. Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu select **Constants** and enter the following constants; then click **OK**.

NAME	EXPRESSION	DESCRIPTION
M	30.3[N*m]	Torque amplitude
R	5[mm]	Radius where torque and force are applied
k	M/(R^4*pi/2)	Constant used in distributed moment calculation
param	1	Parameter used to control the load cases
F	2.95[kN]	Maximum bending force
A	pi*R^2	Area of surface where force is applied

2 Select **Model Settings** from the **Physics** menu to open the **Model Settings** dialog box.

3 Select **MPa** from the **Base unit system** list to use mm as length scale and MPa as stress unit. Click **OK**.

GEOMETRY MODELING

Create the geometry by drawing a 2D plane and rotate it.

1 Select **Work Plane Settings** from the **Draw** menu to open the **Work Plane Settings** dialog box.

2 Click **OK** to create a 2D work plane in the *xy*-plane which is the default settings.

3 Select **Axes/Grid Settings** from the **Options** menu and give axis and grid settings according to the following table. On the **Grid** page, clear the **Auto** check box to enter the grid spacing.

AXIS		GRID	
x min	-5	x spacing	1
x max	60	Extra x	
y min	-15	y spacing	1
y max	15	Extra y	

4 Click the **Line** button on the Draw toolbar. Click the left mouse button at (0, 0), then move the mouse to (0, 10) and click the left mouse button again.

5 Move the mouse to (25, 10) and click the left mouse button again.

6 Move the mouse to (25, 8) and click the left mouse button.

7 Click the **2nd Degree Bézier Curve** toolbar button.

8 Move the mouse to (25, 5) and click the left mouse button.

9 Move the mouse to (28, 5) and click the left mouse button.

10 Click the **Line** button on the Draw toolbar.

11 Move the mouse to (50, 5) and click the left mouse button.

12 Move the mouse to (50, 0) and click the left mouse button.

13 Click the right mouse button to form a 2D solid.

14 Select **Revolve** from the **Draw** menu to open the **Revolve** dialog box.

15 Select **Axis direction through: Second point**.

16 Enter **x** `1` **y** `0` as the coordinates of the **Second point**.

17 Click **OK** to close the **Revolve** dialog box and create the shaft.

PHYSICS SETTINGS

Boundary Conditions

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

2 Specify boundary settings according to the following table:

SETTINGS	BOUNDARIES 1, 3, 5, 7	BOUNDARIES 21–24
Page	Constraint	Load
Constraint condition	Fixed	
F_x		0
F_y		`-z*k*(param>1.5)`
F_z		`y*k*(param>1.5)-F/A*(param<1.5)`

Subdomain Settings

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

2 Specify subdomain settings according to the following table:

SETTINGS	SUBDOMAIN 1
Page	Material
Material model	Isotropic
E	`100[GPa]`
ν	0

MESH GENERATION

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

2 On the **Subdomain** page, type `1.2` in the **Element growth rate** edit field.

3 On the **Boundary** page, select Boundaries 11, 12, 14, and 16.

4 Enter `1` in the **Maximum element size** edit field. Click **OK**.

COMPUTING THE SOLUTION

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

2 Select **Parametric** in the **Solver** list.

3 Specify `param` in the **Parameter name** edit field.

4 Specify `1,2` in the **Parameter values** edit field.

5 Select **Conjugate gradients** from the **Linear system solver** list. Click **OK**.

6 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

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

2 Select **1** from the **Parameter value** list on the **General** page to plot the von Mises stress distribution from the bending.

3 Select **2** from the **Parameter value** list to plot the von Mises stress distribution from torsion.

Fatigue Analysis

This section describes how to solve the fatigue problem.

You have already solved the static bending and torsion load cases.

- Select **File>Export>FEM Structure as 'fem'** from the **File** menu to export the FEM structure containing the static load cases to the command line.
- Run the script `shaft_with_fillet_fatigue.m` by typing `shaft_with_fillet_fatigue` and pressing Return.

The script `shaft_with_fillet_fatigue.m` is shown below.

```
% Compute all components of the stress tensor
[sxx, syy, szz, sxy, syz, sxz] = posteval(fem,'sx_smsld','sy_smsld',...
                                            'sz_smsld',...
                                            'sxy_smsld','syz_smsld',...
                                            'sxz_smsld',...
                                        'edim',2,'solnum','all','cont','on');
sigma = zeros(6,size(sxx.d,2),size(sxx.d,1));
sigma(1,:,:)=sxx.d';
```

```
sigma(2,:,:)=syy.d';
sigma(3,:,:)=szz.d';
sigma(4,:,:)=sxy.d';
sigma(5,:,:)=syz.d';
sigma(6,:,:)=sxz.d';

% Fatigue data for the material
params.f = 440;
params.k = 0.23;

% Combine the basic load cases to form the fatigue load cycle
lccomb = [0 0; 1 1; 1 -1];

% Resolution when searching for the critical plane
angle_step = 6; % degrees

% Compute the fatigue damage
[resval, sigma_max, delta_tau, worst_ind, s_history]= hcfmultiax(sigma, ...
               'lccomb', lccomb, 'anglestep', angle_step, 'params', params);

% Copy a postdata struct from the stress evaluation and assign the damage data
% to the value field (d)
findley_res = sxx;
findley_res.d(1,:) = resval;
figure(1);

% Plot the fatigue damage
postdataplot(findley_res,...
             'scenelight','on',...
             'campos',[244,-161,77],...
             'camtarget',[24.5,0,0],...
             'camva',9.5,...
             'title','Fatigue utilization factor');

% Plot the maximum normal stress in the critical plane
sigma_max_res = sxx;
sigma_max_res.d(1,:) = sigma_max;
figure(2);
postdataplot(sigma_max_res,...
             'scenelight','on',...
             'campos',[244,-161,77],...
             'camtarget',[24.5,0,0],...
             'camva',9.5,...
             'title','Maximum normal stress');

% Plot the shear stress in the critical plane
delta_tau_res = sxx;
delta_tau_res.d(1,:) = delta_tau;
figure(3);
postdataplot(delta_tau_res,...
             'scenelight','on',...
             'campos',[244,-161,77],...
             'camtarget',[24.5,0,0],...
             'camva',9.5,...
             'title','Shear stress amplitude');
```

```
% Plot the stress history for all stress components at the point with
% maximum fatigue damage
x = 1:(size(s_history,2));
figure(4);
if isscript
  plot(x,s_history,'linestyle','cycle');
else
  plot(x,s_history);
end
legend('sxx','syy','szz','sxy','syz','sxz')
title('Stress history in critical point')
```

Frame with Cutout

Introduction

In a beam of a load-carrying frame, there is a cutout which causes stress concentrations.

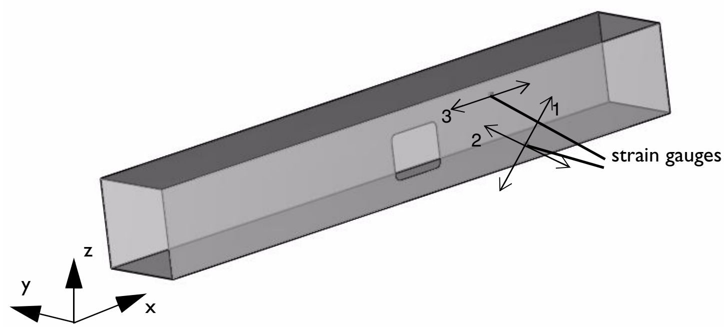


Figure 9-7: Geometry of the beam with cutout showing the location of the strain gauges. The arrows indicate the measurement direction.

Concern arises that fatigue cracks might occur at the cutout as the service loads are increased. Since the loading is random in nature, it needs to be measured before a fatigue analysis can be carried out. The measurements are performed during a few representative service cycles by three strain gauges attached at locations indicated in Figure 9-7.

Based on these measurements, you can perform a fatigue analysis of the beam. First build and solve a static model of the beam, to calculate, from the strain measurements, the stresses at the critical locations around the cutout. You can then use a built-in function to perform a rainflow count on the stress history. By this, load cycles are grouped together based on amplitude and mean stress into groups or bins. Finally, you calculate the accumulated fatigue damage by considering the contribution from each bin.

Note: The fatigue analysis for this model requires COMSOL Script or MATLAB.

Model Definition

Model the beam using shell elements. Fix one end and apply loads to the other end. Assume that the stress distribution in the beam can be obtained by combining together three separate load cases, representing torsion and bending around the two main axes. Solve the model, using the parametric solver, for these three basic load cases.

Use the FE model to compute a relation between the measured strains and the corresponding stresses. The relation between the stress at location i and the strain measured by the gauge at location k can be written as

$$\sigma_i(t) = \sum_{k=1}^{3} q_{ik} \cdot \varepsilon_k(t) \tag{9-1}$$

To find the values of the coefficients, q_{ik}, use the results from the three basic load cases, solving a linear system of equations based on Equation 9-1.

MATERIAL

- Elastic data: Isotropic with E = 200 GPa, ν = 0.3.
- Fatigue data: Iron Alloy 4340 (UNS G43400), three S-N curves for three different R values, extracted from the COMSOL Multiphysics Material Library using the function `matlibfatigue`. You can do this by running the script `extractSNCurvesFromMatlib`. If you do not have the Material Library product the functions have already been extracted for you, and have the following names:
  - `sn_mat1_r_min1` ($R = -1$)
  - `sn_mat1_r_043` ($R = 0.43$)
  - `sn_mat1_r_0` ($R = 0$)

CONSTRAINTS

Fix the edges at one end of the beam.

LOADS

- Moment around the x-axis (torsional load): a constant tangential traction of 100 MPa is applied to each of the four edges.

- Moment around the y-axis (bending load): constant normal stress of +100 MPa on one “flange” constant normal stress of – 100 MPa on the other, and a normal stress along the two “webs” that varies linearly with z.
- Moment around the z-axis (bending load): constant normal stress of +100 MPa on one “flange” constant normal stress of – 100 MPa on the other, and a normal stress along the two “webs” that varies linearly with y.

THE FINITE ELEMENT MESH

Due to the highly nonlinear relation between stress and fatigue life, it is necessary to resolve the stress peaks in detail in a fatigue life analysis. Therefore, use a fine mesh around the fillets in the cutout to study the local stress concentration in this region.

Since high-quality results are also required at the locations of the strain gauges, add two extra points to the geometry in order to control the mesh there. Two of the strain gauges are placed at 45 degrees from the beam axis, see Figure 9-7. For these, you can create a correspondingly rotated local coordinate system to measure the strains.

Results and Discussion

Figure 9-8 shows the von Mises stress for the torsional load case. For this load case, the effect of the cutout is dramatic, with a stress concentration factor that exceeds 10 at the fillets.

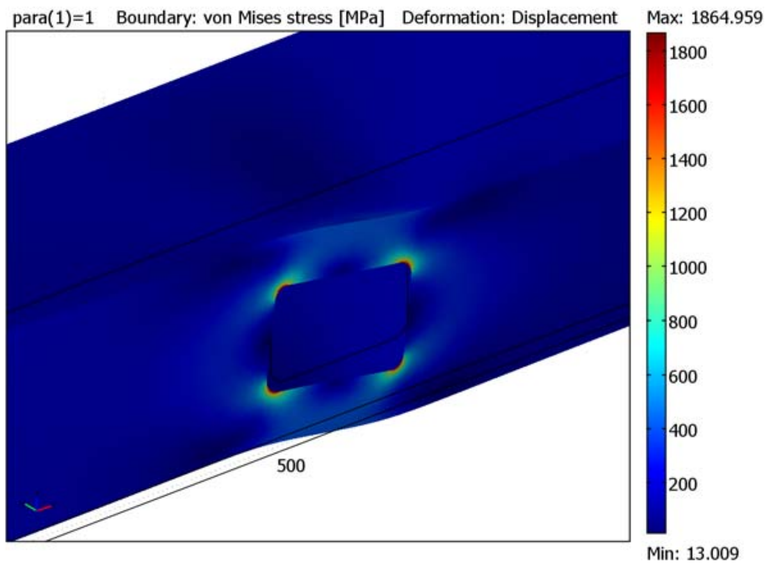


Figure 9-8: von Mises stress around the cutout for the torsional load case.

Figure 9-9 shows the accumulated damage for the evaluation points along the four fillets. For the upper right and lower left corners the damage is close to 1, which means that the material almost reached the allowable number of load cycles until fatigue damage occurs.

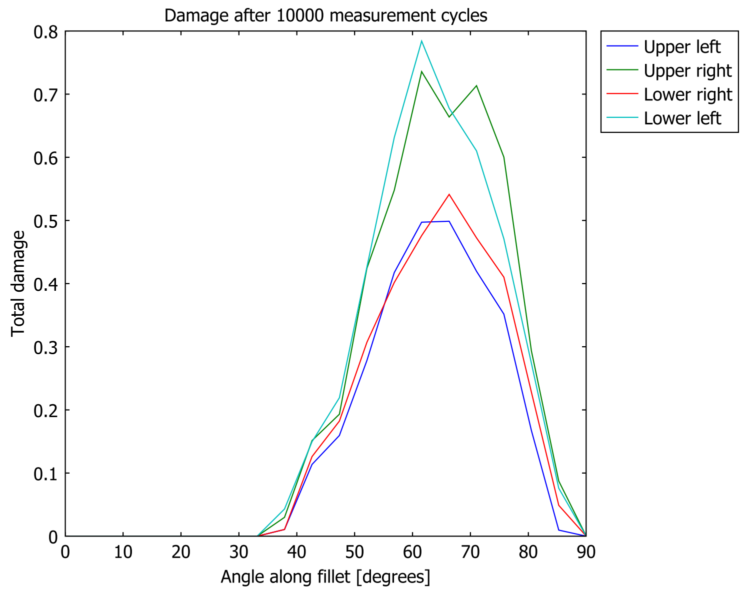


Figure 9-9: Accumulated damage along the four fillets around the cutout.

By plotting the damage distribution and cycle count for the stress bins, you can study how each stress group contributes to the fatigue damage accumulation. In Figure 9-10, you can see that it is almost exclusively certain high loads that contribute to the total damage at the most critical point. However, these high loads only occur once or twice each in the measured load history, as you can see in the cycle count plot for the stress bins, Figure 9-11.

The results illustrate some of the difficulties when performing fatigue analysis. The high stress values can be explained by scatter in the measurements, in which case the accumulated damage value may be too high. Additional measurements during a longer service period would clarify if this is the case, before taking any type of decision on how to modify the load-carrying frame.

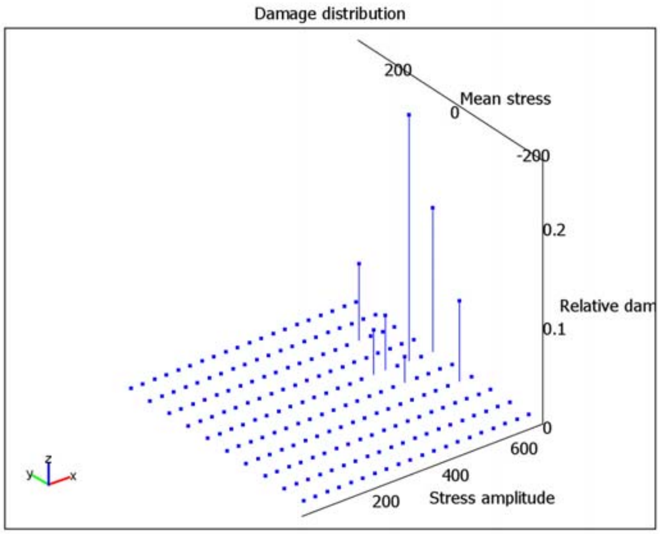


Figure 9-10: Damage distribution as a function of stress bins, characterized by an amplitude and mean stress, for the critical point, which is at the lower left fillet.

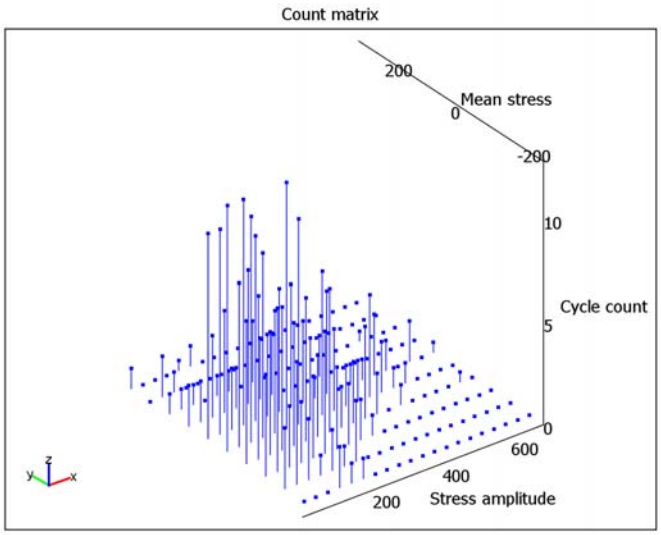


Figure 9-11: Cycle count for each bin, characterized by a stress amplitude and mean stress, for the critical point, which is at the lower left fillet.

Modeling in COMSOL Multiphysics

First, build and solve the model with the three basic load cases of a torsional load and bending load in two directions.

For the fatigue analysis, export the FEM structure to COMSOL Script or MATLAB, where you can carry out the following steps:

- Based on the stresses and strains obtained from the solution of the static problem, compute coefficients, q_{ik}, that appear in Equation 9-1.
- Use these coefficients to calculate the stress history for the locations of interest (at the fillets of the cutout) from the measured strain history.
- With the `rainflow` function, identify groups of load cycles with similar amplitude and mean stress and group these together into bins (rainflow counting). Use 10 groups for the mean stress and 20 groups for the stress amplitude, which results in 200 bins.

- Calculate the accumulated fatigue damage for the binned stress history by using the `fatiguedamage` function.

Model Library path: `Structural_Mechanics_Module/Fatigue/frame_with_cutout`

Modeling Using the Graphical User Interface

This section describes how to solve the three unit load cases using COMSOL Multiphysics.

MODEL NAVIGATOR

1 On the **New** page, select **3D** as **Space dimension**.
2 From the **Application Modes** list, select **Structural Mechanics Module>Shell>Parametric analysis**.
3 Click **OK**.

OPTIONS AND SETTINGS

1 Select **Model Settings** from the **Physics** menu to open the **Model Settings** dialog box.
2 Select **MPa** from the **Base unit system** list to use mm as length scale and MPa as stress unit. Click **OK**.

GEOMETRY MODELING

1 Click the **Block** button on the Draw toolbar to open the **Block** dialog box.
2 Select **Face** in the **Style** area.
3 Enter `1100` as the length in the x direction in the **X** edit field in the **Length** area.
4 Enter `154` as the length in the y and z directions in the **Y** and **Z** edit fields in the **Length** area. Click **OK**.
5 Click **Zoom Extents** on the Main toolbar.
6 Select the block and click the **Split Object** toolbar button to split the block in faces.
7 Select face `F2`, `F3`, and `F5` and press the **Delete** key on your keyboard.
8 Select **Work Plane Settings** from the **Draw** menu to open the **Work Plane Settings** dialog box.

9 Select **z-x** in the **Plane** area and click **OK** to create a 2D work plane in the global *zx*-plane.

10 Select **Specify Objects>Rectangle** from the **Draw** menu to open the **Rectangle** dialog box.

11 Enter 154 as **Width** and 1100 as **Height** and click **OK** to create a rectangle.

12 Select **Specify Objects>Rectangle** from the **Draw** menu to open the **Rectangle** dialog box.

13 Enter 60 as **Width**, 100 as **Height**, 47 as **x** and 500 as **y** and click **OK** to create an additional rectangle.

14 Select **Specify Objects>Rectangle** from the **Draw** menu to open the **Rectangle** dialog box.

15 Enter 80 as **Width**, 80 as **Height**, 37 as **x** and 510 as **y** and click **OK** to create the last rectangle.

16 Select **Specify Objects>Circle** from the **Draw** menu to open the **Circle** dialog box.

17 Enter 10 as **Radius**, 47 as **x** and 510 as **y** and click **OK** to create a circle.

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

19 Click the circle and press Ctrl+C to copy the circle.

20 Click the **Array** toolbar button to make an array copy of the circle.

21 Enter 60 and 80 as **x** and **y Displacement** in the **Array** dialog box**.**

22 Enter 2 as the **x** and **y Array size** and click **OK** to create the circles.

23 Select the circles and the small rectangles and click the **Union** toolbar button.

24 Select both the large rectangle and the rectangle with the rounded corners and click the **Difference** toolbar button to cut out the small rectangle from the large rectangle.

25 Select **Embed** from the **Draw** menu to open the **Embed** dialog box, click **OK** to close the dialog box and embed the rectangle with the cut-out into the 3D geometry.

26 Select **Point** from the **Draw** menu to open the **Point** dialog box. Enter, 850, 77, 0 as **Coordinates**, then click **OK** to close the **Point** dialog box and create the new point.

27 Select **Point** from the **Draw** menu to open the **Point** dialog box. Enter, 850, 154, 77 as **Coordinates**, then click **OK** to close the **Point** dialog box and create the second point.

PHYSICS SETTINGS

Boundary Conditions

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

2 Specify boundary settings according to the following table:

SETTINGS	BOUNDARIES 1–4	BOUNDARIES 3, 4	BOUNDARY 2
Page	Material	Postprocessing	Postprocessing
E	200[GPa]		
ν	0.33		
thickness	6		
Evaluate at		Bottom of shell	
xlocal			1[m] 1[m] t1z[m]

The coordinate system for postprocessing is specified to coincide with the directions of the strain gauges.

Edge Settings

1 Select **Edge Setting**s from the **Physics** menu.

2 Specify edge settings according to the following tables:

SETTINGS	EDGES 1, 2, 4, 6	EDGES 17–20
Page	Constraint	Load
Constraint condition	Fixed	
Load definition		Load is defined as force/area and moment/area using the thickness

SETTINGS	EDGE 17
Page	Load
F_x	100[MPa]/77[mm]*(z-77)*(para>1.5)*(para<2.5)+(para>2.5)*100[MPa]
F_z	-100[MPa]*(para<1.5)

SETTINGS	EDGE 18
Page	Load
F_x	-100[MPa]*(para>1.5)*(para<2.5)-(para>2.5)*100[MPa]/77[mm]*(y-77)
F_y	100[MPa]*(para<1.5)

SETTINGS	EDGE 19
Page	Load
F_x	100[MPa]*(para>1.5)*(para<2.5)-(para>2.5)*100[MPa]/77[mm]*(y-77)
F_y	-100[MPa]*(para<1.5)

SETTINGS	EDGE 20
Page	Load
F_x	`100[MPa]/77[mm]*(z-77)*(para>1.5)*(para<2.5)-(para>2.5)*100[MPa]`
F_z	`100[MPa]*(para<1.5)`

MESH GENERATION

1 From the **Mesh** menu, choose **Free Mesh Parameters** to open the **Free Mesh Parameters** dialog box.

2 Click the **Global** page and select **Extra fine** from the **Predefined mesh sizes** list.

3 Click the **Custom mesh size** radio button.

4 Enter `20` in the **Maximum element size** edit field.

5 Enter `1.2` in the **Element growth rate** edit field.

6 Enter `0.2` in the **Mesh curvature factor** edit field.

7 Click the **Edge** page and select Edges 10, 11, 14, and 15.

8 Enter `0.5` in the **Maximum element size** edit field.

9 Enter `1.1` in the **Element growth rate** edit field.

10 Click the **Point** page and select Points 13 and 14.

11 Enter `1` in the **Maximum element size** edit field.

12 Enter `1.1` in the **Element growth rate** edit field.

13 Click the **Remesh** button and then click **OK**.

COMPUTING THE SOLUTION

1 From the **Solve** menu, choose **Solver Parameters**.

2 In the **Parameter name** edit field, type `para`.

3 In the **Parameter values** edit field, type `1:3`.

4 Click **OK**.

5 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 From the **Postprocessing** menu, choose **Plot Parameters**.

2 Select **1** from the **Parameter value** list to plot the von Mises stress distribution from the bending. Click **Apply**.

3 Select **2** from the **Parameter value** list to plot the von Mises stress distribution from torsion. Click **OK**.

Fatigue Analysis

This section describes how to solve the fatigue problem.

You have already solved the three unit load cases.

- Select **File>Export>FEM Structure as 'fem'** from the **File** menu to export the FEM structure containing the unit load cases to the command line.
- Run the script `extractSNCurvesFromMatlib.m` by typing `extractSNCurvesFromMatlib` and pressing Enter. The script extract three S-N functions for the Steel Alloy 4340 (UNS G43400) material. The `extractSNCurvesFromMatlib` script requires the Material Library. The S-N functions needed in this model example have already been extracted from the Material Library, so you do not need to run the script in order to continue with the analysis.

The script `extractSNCurvesFromMatlib.m` is shown below.

```
%   Material Steel alloy UNS G43400

%   Uniaxial test at 293K and 2000-2500 cpm,
%   R=-1, un-notched rolled bar, 1.125 diameter, air melted,
%   0.4 inch diameter test sample, hand polished to RMS 5-10,
%   longitudinal direction, UTS=200ksi

found = matlibfatigue('material', '4340 (UNS G43400)', 'phase',...
                     'UTS 200 Ksi - 293K', 'funcname', 'sn_mat1_r_min1',...
                      'ori', 'R=-1; unnotched', 'rvalue', -1);
    if (~found)
      error(['material 4340 (UNS G43400), UTS 200 Ksi - 293K, R=-1; ',...
             'unnotched: not found in material library']);
    end

%   Uniaxial test at 293K and 2000-2500 cpm,
%   R=0.43, un-notched rolled bar, 1.125 diameter, air melted,
%   0.4 inch diameter test sample, hand polished to RMS 5-10,
%   longitudinal direction, UTS=200ksi

found = matlibfatigue('material', '4340 (UNS G43400)', 'phase',...
                      'UTS 200 Ksi - 293K', 'funcname', 'sn_mat1_r_043',...
                      'ori', 'R=0.43; unnotched', 'rvalue', 0.43);
    if (~found)
      error(['material 4340 (UNS G43400), UTS 200 Ksi - 293K, R=0.43; ',...
             'unnotched: not found in material library']);
    end

%   Uniaxial test at 293K and 2000-2500 cpm,
%   R=0, un-notched rolled bar, 1.125 diameter, air melted,
%   0.4 inch diameter test sample, hand polished to RMS 5-10,
%   longitudinal direction, UTS=200ksi
```

```
found = matlibfatigue('material', '4340 (UNS G43400)', 'phase',...
                      'UTS 200 Ksi - 293K', 'funcname', 'sn_mat1_r_0',...
                      'ori', 'R=0; unnotched', 'rvalue', 0);
    if (~found)
      error(['material 4340 (UNS G43400), UTS 200 Ksi - 293K, R=0; ',...
             'unnotched: not found in material library']);
    end
```

- Run the script `frame_with_cutout_fatigue.m` by typing `frame_with_cutout_fatigue` and pressing Enter.

The script `frame_with_cutout_fatigue.m` is shown below.

```
eps_pos = [850; 77; 0];  %Location of first strain gauge
ex_gauge_p1 = postinterp(fem,'exl_smsh',eps_pos,'solnum','all','edim',2);
ey_gauge_p1 = postinterp(fem,'eyl_smsh',eps_pos,'solnum','all','edim',2);

eps_pos = [850; 154; 77]; %Location of second strain gauge
ex_gauge_p2 = postinterp(fem,'ex_smsh',eps_pos,'solnum','all','edim',2);

% X and Z coordinates for all four fillet radii centres
x0r = 600-10;
x0l = 500+10;
z0u = 77+40-10;
z0d = 77-40+10;

r = 10.002; %Fillet radius (with a little roundoff tolerance)

nstep = 20; % Number of evaluation steps along each fillet

s1 = zeros(3, nstep*4); % Storage for max principal stress
s3 = zeros(3, nstep*4); % Storage for min principal stress

% For all points along the fillets, get the pricipal stresses
for k = 1:nstep
  phi =pi/2/(nstep-1)* (k-1);
  x = x0l - r*cos(phi);
  z = z0u + r*sin(phi);
  sig_pos = [x; 0; z];
  [s1(:,k), s3(:,k)] = postinterp(fem,'s1_smsh','s3_smsh', ...
    sig_pos,'solnum','all','edim',2);
  x = x0r + r*cos(phi);
  z = z0u + r*sin(phi);
  sig_pos = [x; 0; z];
  [s1(:,k+nstep), s3(:,k+nstep)] = postinterp(fem,'s1_smsh', ...
    's3_smsh',sig_pos,'solnum','all','edim',2);
  x = x0r + r*cos(phi);
  z = z0d - r*sin(phi);
  sig_pos = [x; 0; z];
  [s1(:,k+2*nstep), s3(:,k+2*nstep)] = postinterp(fem,'s1_smsh', ...
    's3_smsh',sig_pos,'solnum','all','edim',2);
  x = x0l - r*cos(phi);
  z = z0d - r*sin(phi);
  sig_pos = [x; 0; z];
  [s1(:,k+3*nstep), s3(:,k+3*nstep)] = postinterp(fem,'s1_smsh', ...
```

```
    's3_smsh',sig_pos,'solnum','all','edim',2);
end

% For the basic load cases, either s1 or s3 will be dominating,
% since the stress state is uniaxial.
% Select the interesting component and store in "stress".
stress = s1.*(abs(s1) > abs(s3)) + s3.*(abs(s1) < abs(s3));

% Compute the "transfer coefficients" q from strain gauges to stress
locations
A = [ex_gauge_p1 ey_gauge_p1 ex_gauge_p2];
q = (A \ stress)';

% Read the measurements from file
load frame_with_cutout_strain_gauge_values.dat
gauge_val=frame_with_cutout_strain_gauge_values';
hist_len = size(gauge_val,2);

% The strain gauge values
figure(1)
plot(1:hist_len,gauge_val(1,:),1:hist_len,gauge_val(2,:),...
     1:hist_len,gauge_val(3,:))
xlabel('time [s]')
ylabel('strain')
title('Strain gauge measurements')

% Compute corresponding stress point histories, transform to SI units
stress_histories = q * gauge_val*1e6;

na = 20; % Number of amplitude bins
nm = 10; % Number of mean stress bins

worst_damage = 0;
for k = 1:nstep*4 % Loop for all stress points
  [range, meanRange, count] = rainflow(stress_histories(k,:), na, nm);
  ampl = range / 2;
  [total_damage(k), damage_dist] = fatiguedamage('amprange', ...
    ampl,'meanrange', ...
    meanRange, 'count',count,'nrepeat', 10000,...
    'sncurves',{'sn_mat1_r_min1'; 'sn_mat1_r_0'; 'sn_mat1_r_043'}, ...
    'rvalue',[-1 0 0.43], 'fatiguelim',[5.0e8 3.8e8 3.0e8]);
  if total_damage(k) > worst_damage
    worst_damage_ind = k;
    worst_damage = total_damage(k);
  end
end

% Create damage plots of critical point
[range, meanRange, count] = rainflow(...
                             stress_histories(worst_damage_ind,:), na, nm);
ampl = range / 2;
da = (ampl(2) - ampl(1)) / na;
dm = (meanRange(2) - meanRange(1)) / nm;

for i = 1:na
```

```
  x(i) = (ampl(1) + (i-0.5) * da) * 1e-6;
end

for i = 1:nm
  y(i) = (meanRange(1) + (i-0.5) * dm) * 1e-6;
end
[worst_damage, worst_damage_dist] = fatiguedamage('amprange',ampl, ...
  'meanrange', meanRange, 'count',count,'nrepeat', 10000, ...
  'sncurves',{'sn_mat1_r_min1'; 'sn_mat1_r_0'; 'sn_mat1_r_043'}, ...
  'rvalue', [-1 0 0.43], 'fatiguelim',[5.0e8 3.8e8 3.0e8]);


figure(2);
stem3(x,y,worst_damage_dist');
title('Damage distribution');
xlabel('Stress amplitude');
ylabel('Mean stress');
zlabel('Relative damage');

figure(3);
stem3(x,y,count');
title('Count matrix');
xlabel('Stress amplitude');
ylabel('Mean stress');
zlabel('Cycle count');

figure(4)
h = plot(90/(nstep-1)*(0:nstep-1), reshape(total_damage,nstep,4));
title('Damage after 10000 measurement cycles');
xlabel('Angle along fillet [degrees]');
ylabel('Total damage');
legend('Upper left','Upper right','Lower right','Lower left');
```

Cylinder with Hole

Introduction

A load carrying component of a structure is subjected to multi-axial cyclic loading during which localized yielding of the material occurs. In this model you perform a low cycle fatigue analysis of the part based on the Smith-Watson-Topper (SWT) model. Due to localized yielding, you can use two methods to obtain the stress and strain distributions for the fatigue law. The first method is an elasto-plastic analysis with linear kinematic hardening, while the second is a linear elastic analysis with Neuber correction for plasticity, based on the Ramberg-Osgood material model.

Model Definition

The cylindrical geometry contains a hole, drilled perpendicularly to its axis. As the structure and loading contains several symmetries, you may model only 1/8 of the cylinder.

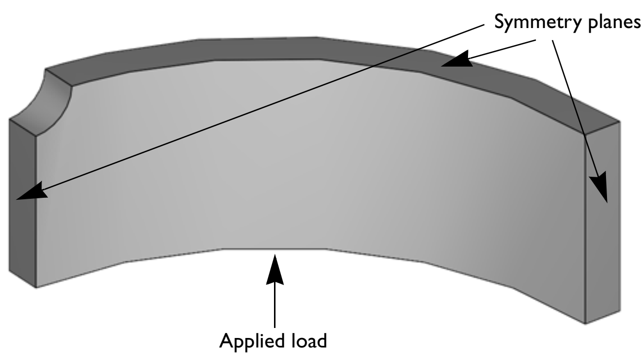


Figure 9-12: Model geometry with constrained and loaded faces.

Model the cylinder using the parametric solver to obtain the stress and strain distributions during a full load cycle. Assume an elasto-plastic material model with linear kinematic hardening. Use the first elastic load step of this model as the basis for the second simplified linear elastic analysis. Since you can apply superposition to obtain the stress distribution for a linear elastic analysis, you can simply multiply the stresses at the first load step with a factor to obtain the stresses at the maximum load.

For the linear elastic fatigue analysis, assume that the local stress state does not depend on the overall elasto-plastic behavior. Account for local plastic strain development by the Neuber correction according to Equation 14-24 to Equation 14-30 in the *Structural Mechanics Module User's Guide*, based on the Ramberg-Osgood model. You can compare this material model with linear kinematic hardening in Figure 9-13.

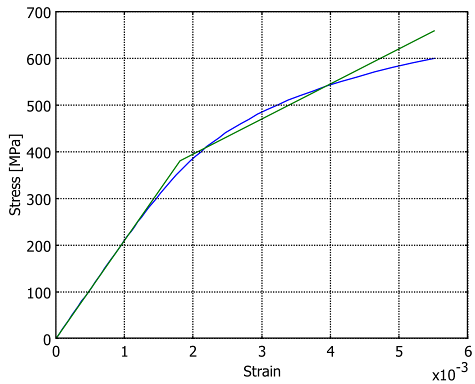


Figure 9-13: Comparison between Ramberg-Osgood and linear kinematic hardening material models.

Note: The fatigue analysis for this model requires COMSOL Script or MATLAB.

MATERIAL PROPERTIES

- Elastic data: Isotropic with E = 210 GPa, ν = 0.3.
- Parameters for the Ramberg-Osgood material model: K' = 1550 MPa, n' = 0.16.
- Kinematic hardening plasticity data:
  - Yield stress 380 MPa
  - Tangent modulus 75 GPa
- Fatigue parameters for the SWT equation:
  - σ_f' = 1323 MPa
  - $b = -0.097$
  - ε_f' = 0.375
  - $c = -0.60$

CONSTRAINTS

Apply symmetry conditions on the three symmetry sections shown in Figure 9-12.

LOADS

Apply two load cycles of a normal stress with an amplitude of 200 MPa and a sinusoidal variation in time (parameter value) to the free bottom face of the cylinder, see Figure 9-12.

Results and Discussion

The von Mises stress when the maximum load is reached for the first time (at a parametric value of 1) in the elasto-plastic analysis is shown in Figure 9-14. Notice that the yield limit is extensively exceeded, and you can therefore expect significant plastic strains.

For a point located on the face of the hole, the greatest stresses appear in the z-direction. In Figure 9-16, you can follow the development of the normal stress and strain in this direction during the two load cycles. As the stress increases, plastic deformation occurs when the yield level is reached. On load reversal, there is an initial elastic deformation followed by new plastic deformation when the effective stress reaches the yield level once more. You can see that after the maximum load is reached for the first time, a stable hysteresis loop develops with subsequent cycles following the original one exactly. This behavior is characteristic of a kinematic hardening model.

The development of the effective plastic strain for the same point appears in Figure 9-15. The regions where effective plastic strain is constant correspond to regions of elastic deformation in Figure 9-16.

To perform the fatigue analysis, you can use the stress and strain distributions from a fully developed hysteresis loop, starting when the maximum load is reached for the first time. The fatigue analysis with the simplified elastic approach is instead based on the stress distribution from the initial linear portion of the stress-strain curve in Figure 9-16.

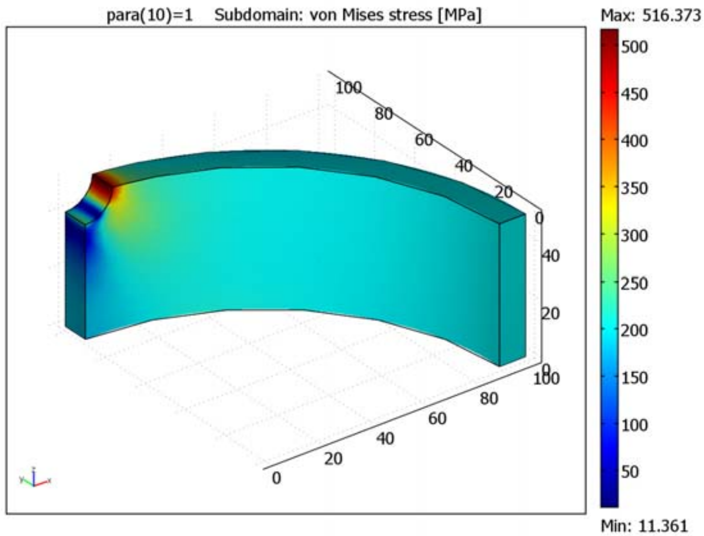


Figure 9-14: von Mises stress level at the first maximum load for the full elasto-plastic analysis.

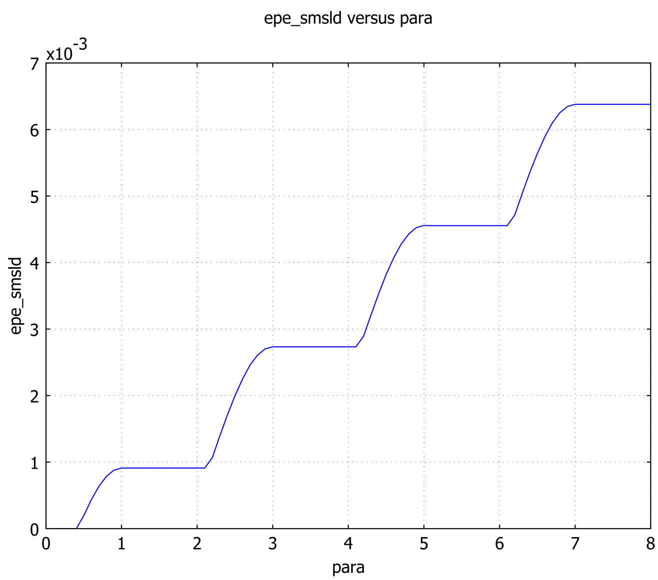


Figure 9-15: Effective plastic strain versus loading parameter. An increase of the parameter value by 4 corresponds to one full load cycle.

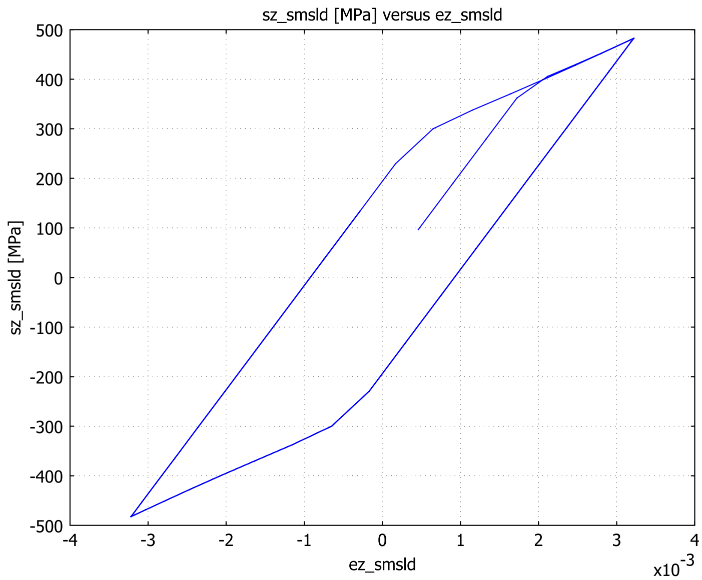


Figure 9-16: σ_z *versus* ε_z *for two full load cycles.*

Plots of the logarithm of the predicted life in number of cycles for the full elasto-plastic and the simplified elastic analyses are shown in Figure 9-17 and Figure 9-18, respectively. An interesting result when predicting the service life of the component is the minimum number on the color scale. As expected, the region with the shortest life is around the hole. The number of load cycles to reach the fatigue limit is predicted to be 6600, when using the full elasto-plastic method, and 4700, when using the simplified analysis.

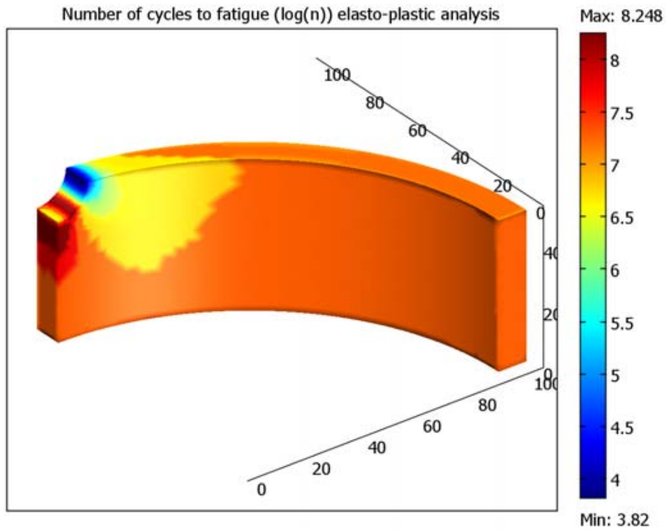


Figure 9-17: Number of cycles to fatigue (log n) for the full elasto-plastic analysis.

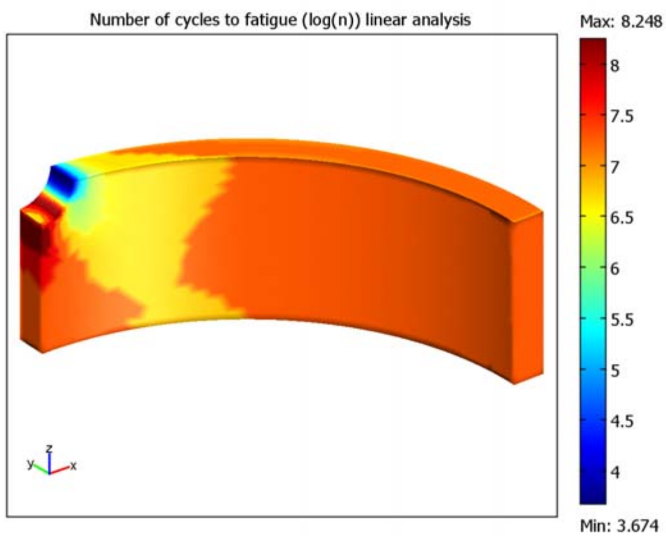


Figure 9-18: Number of cycles to fatigue (log n) for the linear analysis.

Modeling in COMSOL Multiphysics

Build and solve the model of the cylinder with elasto-plastic analysis. Create a finer mesh around the hole to properly resolve the stress distribution in this region. Export the FEM structure with the solved model to either COMSOL Script or MATLAB, where you can continue the modeling.

To carry out a fatigue analysis using a load cycle from the elasto-plastic analysis, do the following:

- Extract stress and strain data from the FEM structure using the `posteval` function. Do this for a fully developed elasto-plastic load cycle, which corresponds to the parametric values of `1` to `5`.
- Calculate the fatigue life by using the `lcfmultiaxpla` function.

Next perform the fatigue analysis based on the linear elastic analysis.

- Use the `posteval` function to compute the principal stresses corresponding to the first parametric value.
- From these results, calculate the principal stresses at the maximum load by multiplying with the appropriate factor.
- Use the `lcfmultiaxlin` function to calculate the fatigue life.

Model Library path: `Structural_Mechanics_Module/Fatigue/cylinder_hole`

Modeling Using the Graphical User Interface

This section describes how to solve the full elasto-plastic analysis using COMSOL Multiphysics and the Structural Mechanics Module.

MODEL NAVIGATOR

1 On the **New** page, select **3D** from the **Space dimension** list and click on **Structural Mechanics Module>Solid, Stress-Strain>Static analysis elasto-plastic material**.

2 Click **OK**.

OPTIONS AND SETTINGS

1 Select **Model Settings** from the **Physics** menu to open the **Model Settings** dialog box.

2 Select **MPa** from the **Base unit system** list to use mm as length scale and MPa as stress unit.

GEOMETRY MODELING

1 Click the **Cylinder** button on the Draw toolbar to open the **Cylinder** dialog box.

2 In the **Cylinder parameters** area, set both the **Radius** and the **Height** to 100.

3 Click **OK** to close the **Cylinder** dialog box and create the cylinder.

By repeating Steps 1–3, draw a second cylinder with a radius of 90 and the same height as the one you just created.

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

5 Press Ctrl+A to select both cylinders, then click the **Difference** button on the Draw toolbar to cut a hole through the large cylinder using the small cylinder.

Next, draw a small cylinder to cut a hole in the wall of the hollow cylinder.

6 Click the **Cylinder** button on the Draw toolbar.

7 In the **Cylinder parameters** area, set the **Radius** to 10 and the **Height** to 100.

8 In the **Axis base point** area, set the **x**, **y**, and **z** components to 0, 0, and 50, respectively.

9 In the **Axis direction vector** area, set the **x**, **y**, and **z** components to 0, 1, and 0, respectively.

10 Click **OK** to create the small cylinder.

11 Select both cylinders and click the **Difference** button on the Draw toolbar.

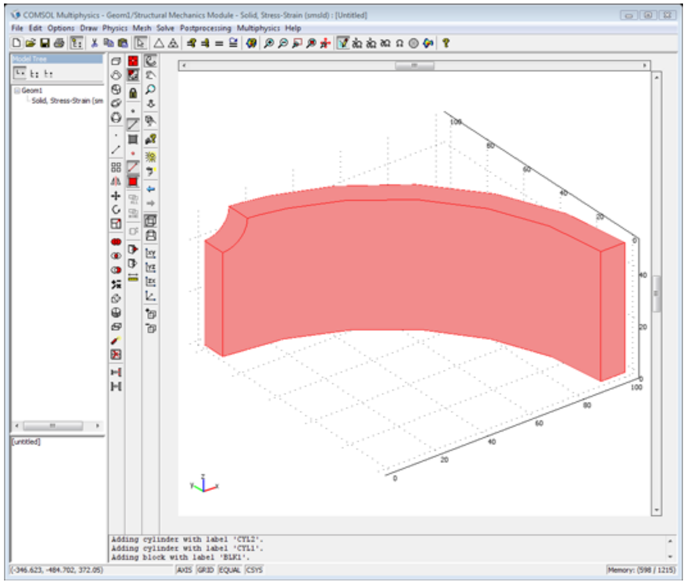
12 Click the **Zoom Extents** button on the Main toolbar.

13 Click the **Block** button on the Draw toolbar to open the **Block** dialog box.

14 In the **Length** area, set **X** to 100, **Y** to 100, and **Z** to 50. Click **OK**.

15 Press Ctrl+A to select both objects.

16 Click the **Intersection** button on the Draw toolbar



PHYSICS SETTINGS

Boundary Conditions

1. Select **Boundary Settings** from the **Physics** menu.
2. Specify boundary conditions and loads according to the following table:

SETTINGS	BOUNDARIES 1, 6, 7	BOUNDARY 3
Page	Constraint	Load
Constraint condition	Symmetry plane	
F_x		0
F_y		0
F_z		`-200*sin(pi*para/2)`

Subdomain Settings

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

2 Specify material properties according to the following table:

SETTINGS	SUBDOMAIN 1
Page	Material
Material model	Elasto-plastic
E	`210[GPa]`
ν	`0.3`

3 Click the **Elasto-plastic material data** button to open the **Elasto-Plastic Material Settings** dialog box.

4 Specify elasto-plastic material data according to the following table:

SETTINGS	SUBDOMAIN 1
Hardening model	Kinematic
E_{Tkin}	`75[GPa]`
σ_{ys}	380

MESH GENERATION

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

2 Click the **Boundary** tab.

3 Select Boundary 4 and enter `1.5` in the **Maximum element size** edit field.

4 Click the **Remesh** button and then click **OK**.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar or select **Solver Parameters** from the **Solve** menu to open the **Solver Parameters** dialog box.

2 In the **Parameter name** edit field, type `para`.

3 In the **Parameter values** edit field, type `0.1:0.1:8`. Click **OK**.

4 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

Reproduce the plot of the von Mises stress in Figure 9-14 with the following steps:

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

2 On the **General** page, clear the **Slice** check box and select the **Subdomain** check box in the **Plot type** area to plot the stresses in the entire domain.

3 Select **1** from the **Parameter value** list to plot the von Mises stress distribution at the first occurrence of the maximum axial loading.

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

Proceed to generate the plots in Figure 9-15 and Figure 9-16 as follows:

1 From the **Postprocessing** menu, select **Domain Plot Parameters**.

2 Click the **Point** tab and select Point 5 from the **Point selection** list.

3 In the **y-axis data** area, type `epe_smsld` in the **Expression** edit field.

4 In the **x-axis data** area, click the option button in front of the **Expression** button.

5 Click the **Expression** button and type `para` in the **Expression** edit field.

6 Click **OK** to close the **X-Axis Data** dialog box.

7 Click **Apply** to generate the plot in Figure 9-15.

8 In the **y-axis data** area, type `sz_smsld` in the **Expression** edit field.

9 In the **x-axis data** area, click the **Expression** button.

10 Type `ez_smsld` in the **Expression** edit field, then click **OK** to close the **X-Axis Data** dialog box.

11 Click **OK** to close the **Domain Plot Parameters** dialog box and create the plot in Figure 9-16.

Fatigue Analysis

This section describes how to solve the fatigue problem.

FULL ELASTO-PLASTIC ANALYSIS

1 Select **File>Export>FEM Structure as 'fem'** from the **File** menu to export the FEM structure containing the full elasto-plastic analysis to the command line.

2 Run the script `cylinder_hole_plastic_fatigue.m` by typing `cylinder_hole_plastic_fatigue` and pressing Return.

The script `cylinder_hole_plastic_fatigue.m` is shown below.

```
% Calculate the stress tensor
[sxx, syy, szz, sxy, syz, sxz] = posteval(fem,'sx_smsld','sy_smsld', ...
  'sz_smsld','sxy_smsld','syz_smsld','sxz_smsld','edim',2, ...
  'solnum',10:50,'cont','on');
sigma = zeros(6,size(sxx.d,2),size(sxx.d,1));
sigma(1,:,:)=sxx.d';
sigma(2,:,:)=syy.d';
sigma(3,:,:)=szz.d';
sigma(4,:,:)=sxy.d';
```

```
sigma(5,:,:)=syz.d';
sigma(6,:,:)=sxz.d';

% Calculate the strain tensor
[exx, eyy, ezz, exy, eyz, exz] = posteval(fem,'ex_smsld','ey_smsld', ...
  'ez_smsld','exy_smsld','eyz_smsld','exz_smsld','edim',2, ...
  'solnum',10:50,'cont','on');
strain = zeros(6,size(exx.d,2),size(exx.d,1));
strain(1,:,:)=exx.d';
strain(2,:,:)=eyy.d';
strain(3,:,:)=ezz.d';
strain(4,:,:)=exy.d';
strain(5,:,:)=eyz.d';
strain(6,:,:)=exz.d';

% Material parameters
para.E = 210000;
para.sigmaf =  1323;
para.b = -0.097;
para.epsf = 0.375;
para.c = -0.60;
angle_step = 15;

% Calculate the fatigue life
[n_cycles_plastic ms mde, swt]= lcfmultiaxpla(sigma, strain, 'anglestep',
...
                                angle_step, 'params',para);

% Plot the number of cycles
res = sxx;
res.d(1,:) = log10(n_cycles_plastic);
figure(1);
postdataplot(res,...
             'scenelight','on',...
             'title','Number of cycles to fatigue (log(n)) elasto-plastic
analysis');

% Plot the maximum normal stress at the critical plane
sres = sxx;
sres.d(1,:) = ms;
figure(2);
postdataplot(sres,...
             'scenelight','on',...
             'title','Maximum normal stress at critical plane');

% Plot the strain amplitude at the critical plane
eres = sxx;
eres.d(1,:) = mde;
figure(3);
postdataplot(eres,...
             'scenelight','on',...
             'title','Strain amplitude at critical plane');

% Plot the swt parameter at the critical plane
swtres = sxx;
```

```
swtres.d(1,:) = swt;
figure(4);
postdataplot(swtres,...
             'scenelight','on',...
             'title','SWT parameter at critical plane');
```

LINEAR ELASTIC ANALYSIS

1 Select **File>Export>FEM Structure as 'fem'** from the **File** menu to export the FEM structure containing the full elasto-plastic analysis results including the linear elastic ones to the command line.
2 Run the script `cylinder_hole_linear_fatigue.m` by typing `cylinder_hole_linear_fatigue` and pressing Return.

The script `cylinder_hole_linear_fatigue.m` is shown below.

```
% Calculate the principal stresses while the material is still
% linear elastic
[s1e, s2e, s3e] = posteval(fem,'s1_smsld','s2_smsld', ...
  's3_smsld','edim',2,'solnum',1,'cont','on');
sig_princ = zeros(3,size(s1e.d,2));
sig_princ(1,:)=s1e.d(1,:);
sig_princ(2,:)=s2e.d(1,:);
sig_princ(3,:)=s3e.d(1,:);

% Divide with the parameter value to get the stresses at the maximum load
% The first step; the load is applied as sin(p*pi/2) with p = 0.1, 0.2 ...
sig_princ = sig_princ * 1 / sin(0.1 * pi / 2);

% Material parameters
para.E = 210000;
para.nu= 0.3;
para.K = 1550;
para.n = 0.16;
para.sigmaf = 1323;
para.b = -0.097;
para.epsf = 0.375;
para.c = -0.60;

% Calculate the number of cycles
n_cycles_linear = lcfmultiaxlin(sig_princ, 'params', para);

% Plot the number of cycles
res = s1e;
res.d(1,:) = log10(n_cycles_linear);
figure(5)
postdataplot(res,...
             'scenelight','on',...
            'title','Number of cycles to fatigue (log(n)) linear analysis');
```

Fatigue Analysis of an Automobile Wheel Rim

Introduction

When designing a lightweight rim for a car it is of utmost importance to avoid fatigue failure. A number of fatigue tests can be performed in order to ascertain that the rim will not fail. Resorting only to tests, however, leads to large costs for prototype tools and a long development cycle, which is easy to avoid by simulating the testing procedure.

One such test is "rolling fatigue." The wheel is pressed against a rotating drum with a force that is given by the intended capacity of the wheel. During one revolution of the wheel, each material point experiences a complex stress history, which adds one cycle to the fatigue damage.

Note: This model requires COMSOL Script or MATLAB.

Model Definition

The wheel rim for this analysis is a ten-spoke example where the design elements of the geometry cause the finite element mesh to become quite large. To reduce the size of the problem, use an iterative solver and make use of symmetry while modeling. The loading on the tire is composed of both the tire pressure and a rotating load (drum load) transferred from the tire to the rim. In this case, only the geometry is symmetric, while the loading (which moves) is not. Because the problem is linear, superposition of load cases is possible. Any load on a symmetric structure can be separated into one symmetric and one antisymmetric load, as the following figure illustrates.

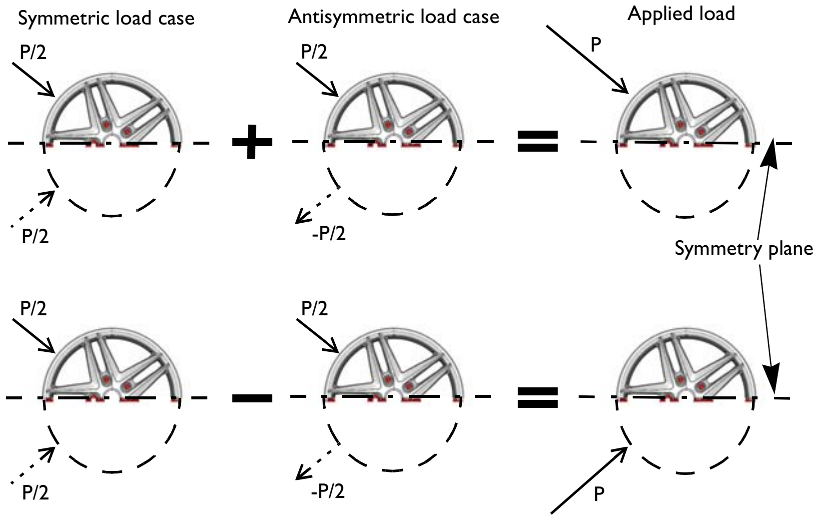
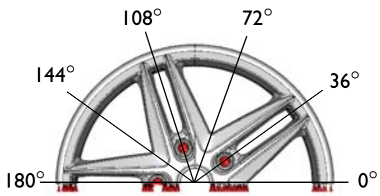


Figure 9-19: Superposition of symmetric and antisymmetric load cases to obtain the stress distribution for a load applied in two locations.

For an applied load at a certain position you can solve the model using one half of the geometry with symmetric and antisymmetric boundary conditions, respectively. Adding and subtracting the solution from these load cases results in the stress distribution for a load applied in two symmetric positions, as outlined in Figure 9-19.

In the analysis you study the problem in a coordinate system fixed to the wheel rim, where the load rotates around the wheel. Assume that the load on the rim extends 30° in each direction from the point of contact. It is reasonable to analyze the stress distribution with the point of contact evenly spread out at ten locations around the rim. For the modeled half of the geometry this is equivalent to six load positions with 36° interval between them as shown in the following figure.



You can model the moving load by letting the parametric solver step through the parameter values, which you can then use in the load expression to control its location and distribution. Use Table 9-1 and Table 9-2 to find each load case by its parameter value and load angle. Combine the load cases according to Table 9-3 to calculate the stress distribution for load positions around the entire wheel. The designations used in the Combination column correspond to load cases listed in Table 9-1 and Table 9-2.

TABLE 9-1: SYMMETRIC LOAD CASES

PARAMETER VALUE	DESCRIPTION	DESIGNATION
1	Drum load 0°	sd1
2	Drum load 36°	sd2
3	Drum load 72°	sd3
4	Drum load 108°	sd4
5	Drum load 144°	sd5
6	Drum load 180°	sd6
7	Tire pressure	st7

TABLE 9-2: ANTISYMMETRIC LOAD CASES

PARAMETER VALUE	DESCRIPTION	DESIGNATION
2	Drum load 36°	ad2
3	Drum load 72°	ad3
4	Drum load 108°	ad4
5	Drum load 144°	ad5

TABLE 9-3: COMBINED LOAD CASES

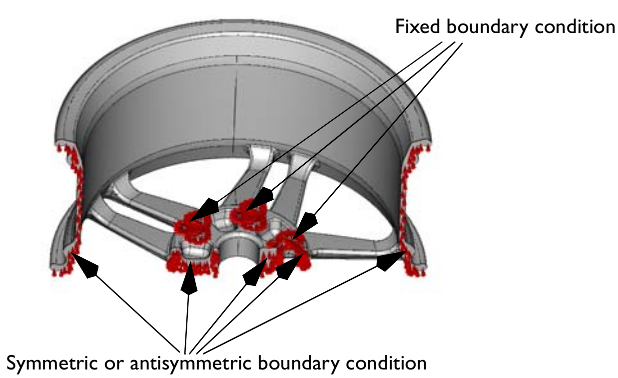
PARAMETER VALUE	DESCRIPTION	COMBINATION
0	Tire pressure + Drum load 0°	st7 + sd1
36	Tire pressure + Drum load 36°	st7 + (sd2 + ad2)/2
72	Tire pressure + Drum load 72°	st7 + (sd3 + ad3)/2
108	Tire pressure + Drum load 108°	st7 + (sd4 + ad4)/2
144	Tire pressure + Drum load 144°	st7 + (sd5 + ad5)/2
180	Tire pressure + Drum load 180°	st7 + sd6
216	Tire pressure + Drum load 216°	st7 + (sd5 - ad5)/2
252	Tire pressure + Drum load 252°	st7 + (sd4 - ad4)/2
288	Tire pressure + Drum load 288°	st7 + (sd3 - ad3)/2
324	Tire pressure + Drum load 324°	st7 + (sd2 - ad2)/2

MATERIAL

- Elastic data: Isotropic with E = 70 GPa, ν = 0.33.
- Fatigue data: The fatigue limit is known for two cases with pure axial loading. For pure tension it is 95 MPa, and for fully reversed loading it is 125 MPa. This gives the Findley parameters f = 84 MPa and k = 0.30 if Equation 14-16 in the *Structural Mechanics Module User's Guide* is applied.

CONSTRAINTS

- A region around each bolt hole is fixed.

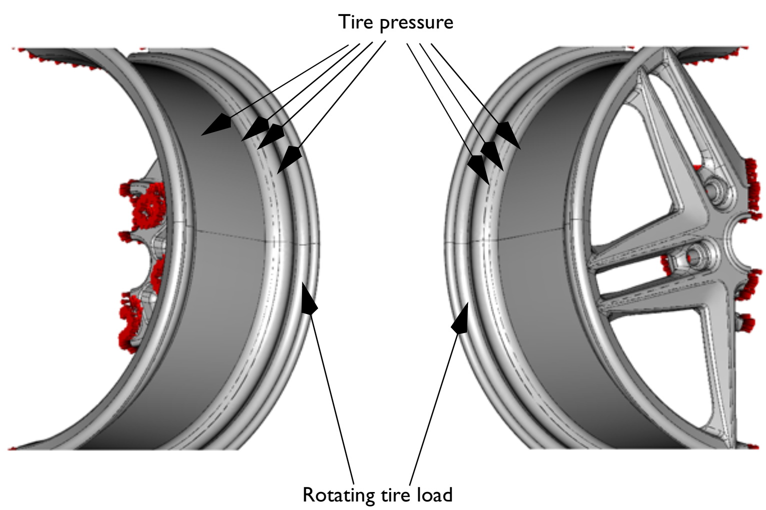


- Symmetric load cases: Symmetry condition (normal displacement fixed) in the symmetry plane.
- Antisymmetric load cases: Antisymmetry condition (transverse displacement fixed) in the symmetry plane.

LOADS

- Tire pressure: The overpressure is 2 bar = 200 kPa.
- Rotating drum load: The total load carried by the wheel corresponds to a weight of 1120 kg. It is applied as a pressure on the rim surfaces where the tire is in contact. Assume that the load distribution in the circumferential direction can be approximated as $p = p_0 \cos(3\vartheta)$, where ϑ is the angle from the point of contact between the drum and the tire. The loaded area thus extends 30° in each direction. The parameter k_{solv} in the following expression is used to control the location of the load:

$$p = -p_0(k_{\mathrm{solv}} < 6)\cos\left[3\,\mathrm{atan}\left(\frac{y}{x}\right) - \frac{\pi(2k_{\mathrm{solv}} - 7)}{10}\right] \cdot$$
$$\cdot \left[\mathrm{atan}\left(\frac{y}{x}\right) < \frac{\pi(3k_{\mathrm{solv}} - 13)}{15}\right] \cdot \left[\mathrm{atan}\left(\frac{y}{x}\right) > \frac{\pi(3k_{\mathrm{solv}} - 8)}{15}\right]$$



THE FINITE ELEMENT MESH

Mesh the rim surface with a fine mesh to resolve all the details, and to avoid inverted and linearized elements, which may influence the fatigue analysis.

The final finite-element mesh consists of approximately 310,000 tetrahedral elements, giving a total number of approximately 1,500,000 degrees of freedom.

Results and Discussion

The von Mises stress for one of the combined load cases with the point of contact for the drum load located at 144° is shown below, illustrating the region for where the rim experiences the maximum stresses.

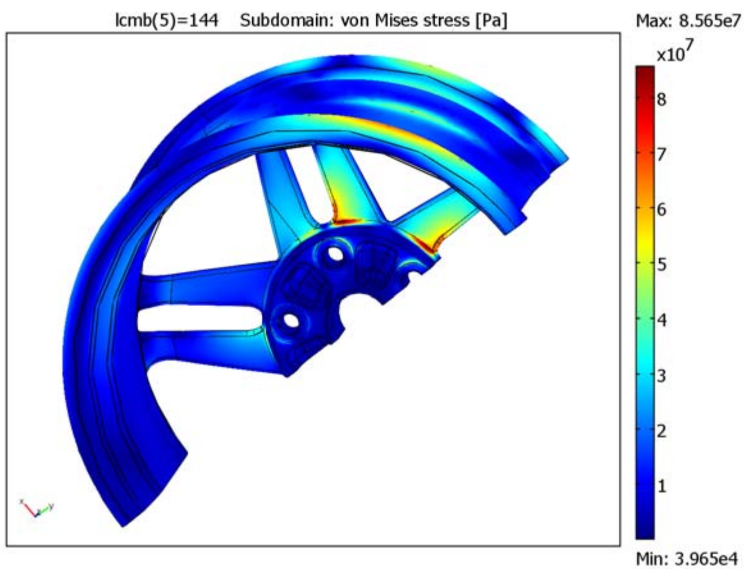


Figure 9-20: von Mises stress distribution for the combined loading with the rotating drum load at 144°.

The computed fatigue utilization factor is shown in Figure 9-21. The maximum value is 0.717. As you can see the critical regions are on the inside where the spokes connect to the mid part of the rim. When comparing with the stress plot in Figure 9-20, note that high stresses will not always lead to a high utilization factor. This is the case for the two edges that run around the rim holding the tire in place.

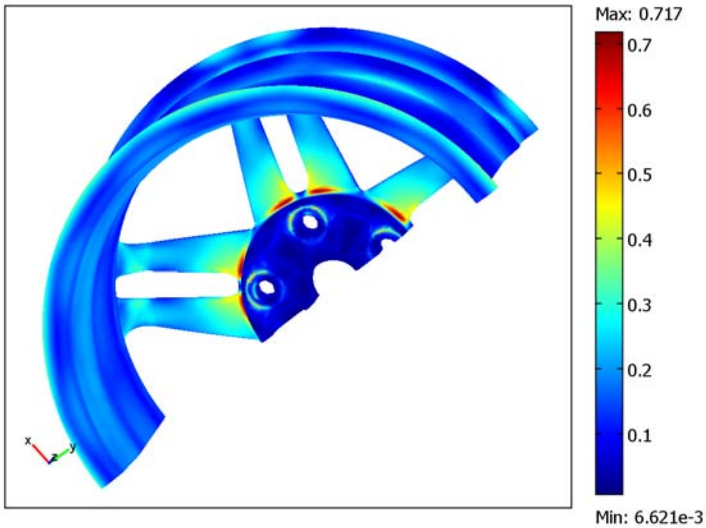


Figure 9-21: Distribution of the fatigue utilization factor.

By looking at the plot in Figure 9-22 you can understand the nonproportional nature of the stress history. Here you can see how the components of the stress tensor change at the most critical point as the load rotates around the wheel. For proportional loading there would only be a scale difference between the curves. But in this case principal directions of the stress tensor change with each step.

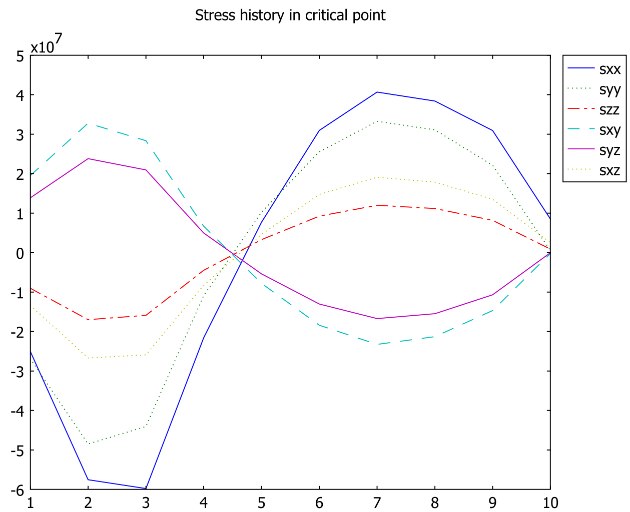


Figure 9-22: Stress history at the location with the highest fatigue utilization factor.

Modeling in COMSOL Multiphysics

Build and solve two models, the first one containing the symmetric load cases with the pressure load and the second one containing the antisymmetric load cases.

To superpose the symmetric and antisymmetric load cases, export the FEM structures from these two models to COMSOL Script, where you can extract and add the solution vectors.

Perform the fatigue analysis using either COMSOL Script or MATLAB, where you can run scripts that analyzes the total stress distribution stored in the FEM structures.

Model Library path: `Structural_Mechanics_Module/Fatigue/wheel_rim_fatigue`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Click the **New** button or start COMSOL Multiphysics to open the **Model Navigator**.
2 Select **3D** from the **Space dimension** list.
3 Open the **Structural Mechanics Module** folder and then **Solid, Stress-Strain**. Select **Parametric analysis**.
4 Click **OK**.

IMPORT OF CAD GEOMETRY

1 On the **File** menu select **Import>CAD Data From File**.
2 In the **Files of type** list select **COMSOL Multiphysics file**.
3 Browse to the `models/Structural_Mechanics_Module/Automotive_Applications` directory located in the COMSOL installation directory and select the file `wheel_rim.mphbin`.
4 Click **Import**.

OPTIONS AND SETTINGS

Constants

1 Select **Options>Constants**.
2 Enter the constants from the following table in the **Constants** dialog box.

NAME	EXPRESSION
`pressure`	`2e5[Pa]`
`tire_load`	`5.194e6[Pa]`

3 Click **OK**.

Visualization Settings

Use a coarse visualization mesh to reduce the time it takes to rendering the geometry when switching views in the user interface:

1 From the **Options** menu, choose **Visualization/Selection Settings**.
2 From the **Visualization mesh** list, select **Coarse**.
3 Click **OK**.

MESH GENERATION

1 Select **Mesh>Free Mesh Parameters**.

2 On the **Boundary** page select boundaries and enter the parameters according to the following table.

BOUNDARIES	MESH CURVATURE FACTOR	MESH CURVATURE CUTOFF
11, 13, 89, 90	0.5	
27, 37, 41, 99, 100, 106, 108, 109, 112, 128, 129, 130, 144, 147, 154, 169, 171, 180, 181, 182, 191, 198, 199, 204, 205, 220, 228, 230, 236, 256, 274, 275, 278, 280, 291, 303, 313, 333, 343, 345, 349, 350, 352, 353, 357, 358, 373, 375, 376, 377, 386, 411, 413, 414, 415, 417, 418, 426, 427, 428, 433, 444	0.4	
32, 60, 73, 186, 314, 332, 340, 385, 397, 406, 412, 441	0.3	
44, 46, 51, 52	0.1	
3, 5, 6, 7, 92, 94, 95, 96, 107, 196, 229, 317, 360, 362, 378, 387, 391, 407, 408, 416, 419, 429, 434	0.4	0.02
2, 4, 16, 17, 84, 85, 97, 98, 197, 371, 401, 432, 452	0.4	0.015
30, 33, 72, 439	0.3	0.02
28, 29, 54, 74, 75, 103, 104, 115, 116, 126, 132, 145, 148, 151, 177, 194, 207, 237, 238, 239, 247, 248, 251, 252, 253, 254, 255, 265, 270, 279, 281, 285, 288, 290, 292, 298, 309, 319, 320, 321, 322, 325, 326, 327, 328, 329, 330, 334, 335, 337, 341, 342, 346, 348, 351	0.3	0.015
35, 43, 57, 70, 102, 114, 121, 136, 137, 138, 146, 152, 158, 164, 166, 167, 184, 185, 192, 201, 203, 206, 211, 222, 233, 244, 261, 272, 286, 295, 299, 300, 302, 304, 379, 398, 409	0.3	0.01

3 Click **OK**.

4 Click the **Mesh All (Free)** button on the Mesh toolbar.

PHYSICS SETTINGS (SYMMETRIC LOAD CASES)

Subdomain Settings

1 On the **Physics** menu select **Subdomain Settings**.
2 In the **Subdomain selection** list select Subdomain 1.
3 On the **Material** page click the **Load** button to open the **Materials/Coefficients Library** dialog box.
4 In the **Materials** list expand **Basic Material Properties** then select **Aluminum**.
5 Click **OK** to apply the material and close the dialog box.
6 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

1 On the **Physics** menu select **Boundary Settings**.
2 On the **Boundaries** page locate the **Boundary selection** list and select Boundaries 56, 156, and 232.
3 On the **Constraint** page select **Fixed** in the **Constraint condition** list box.
4 From the **Boundary selection** list select Boundaries 1, 26, 48, 64, and 76.
5 On the **Constraint** page select **Symmetry Plane** in the **Constraint condition** list box.
6 From the **Boundary selection** list select Boundaries 10–15, 18, 19, 22, 81-83, 86, 87, 89-91, and 93.
7 On the **Load** page make sure that **Distributed load** is selected from the **Type of load** list box.
8 Select **Tangent and normal coord. sys. (t_1,t_2,n)** from the **Coordinate system** list box.
9 Enter `-pressure*(load_param>6)` in the **F_n** edit field.
10 From the **Boundary selection** list select Boundaries 6, 7, 92 and 94.
11 On the **Load** page make sure that **Distributed load** is selected from the **Type of load** list box.
12 Select **Tangent and normal coord. sys. (t_1,t_2,n)** from the **Coordinate system** list box.
13 In the **F_n** edit field enter

```
-tire_load*(load_param<7)*cos(3*(atan2(y,x)-pi/
10*(2*load_param-7)))*(atan2(y,x)>pi/
15*(3*load_param-13))*(atan2(y,x)<pi/15*(3*load_param-8))
```

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

COMPUTING THE SOLUTION (SYMMETRIC LOAD CASES)

1 Click the **Solver Parameters** button.

2 Select **Parametric** in the **Solver** list.

3 On the **General** page, type `load_param` in the **Parameter name** edit field and the vector `1:7` in the **Parameter values** edit field.

4 From the **Linear system solver** list select **Conjugate gradients**.

5 From the **Preconditioner** list select **Geometric multigrid**.

6 Click the **Settings** button.

7 In the dialog box that opens select **Linear system solver>Preconditioner>Coarse solver**.

8 From the **Coarse solver** list select **SPOOLES**.

9 Click **OK**.

10 Click **OK**.

11 Click the **Solve** button on the Main toolbar.

SAVING THE SYMMETRIC LOAD CASES

After the problem is solved you can save the model in case you want to open and examine the solution to the symmetric load cases. If you do not want to save the model continue with the next section, "Exporting the Symmetric Load Cases to COMSOL Script."

1 Select **File>Save**.

2 Navigate to the directory of your choice.

3 In the **File name** edit field enter `wheel_rim_fatigue_sym.mph`.

4 Click **Save**.

A solved model with the symmetric load cases is available in the COMSOL Model Library root directory under `Structural_Mechanics_Module/Fatigue/wheel_rim_fatigue_sym.mph`.

EXPORTING THE SYMMETRIC LOAD CASES TO COMSOL SCRIPT

You can export the model to COMSOL Script as an FEM structure containing the geometry, mesh, physics settings, as well as the solution. From the FEM structure you can then extract the solution vector, which you can add to the solution vector of the antisymmetric load cases.

1 Select **File>Export>FEM Structure**.

2 In the dialog box that opens enter `fem_sym_cases` in the **Variable name for FEM structure** edit field.

3 Click **OK**. COMSOL Script opens automatically.

4 Switch back to COMSOL Multiphysics before continuing with the next step.

PHYSICS SETTINGS (ANTISYMMETRIC LOAD CASES)

On the symmetry plane of the geometry you need to change the boundary condition to antisymmetry before computing the antisymmetric load cases. Also, you can skip the last parameter step, since it corresponds to the pressure load.

Boundary Conditions

1 On the **Physics** menu select **Boundary Settings**.

2 In the **Boundary selection** list select Boundaries 1, 26, 48, 64, and 76.

3 On the **Constraint** page select **Antisymmetry Plane** in the **Constraint Condition** list box.

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

COMPUTING THE SOLUTION (ANTISYMMETRIC LOAD CASES)

1 Click the **Solver Parameters** button on the Main toolbar.

2 From the **Solver** list, select **Parametric**.

3 On the **General** page, change the vector in the **Parameter values** edit field to `2:6`.

4 Click **OK**.

5 Click the **Solve** button on the **Main** toolbar.

SAVING THE ANTISYMMETRIC LOAD CASES

Carry out the following steps to save the model at this stage. If you do not want to save the model continue with the next section, "Exporting the Antisymmetric Load Cases to COMSOL Script."

1 Select **File>Save As**.

2 Navigate to the directory of your choice.

3 In the **File name** edit field enter `wheel_rim_fatigue_asym.mph`.

4 Click **Save**.

A solved model with the antisymmetric load cases is available in the COMSOL Model Library root directory under `Structural_Mechanics_Module/Fatigue/wheel_rim_fatigue_asym.mph`.

EXPORTING THE ANTISYMMETRIC LOAD CASES TO COMSOL SCRIPT

When you have solved the antisymmetric model, export the solution to COMSOL Script.

1 Select **File>Export>FEM Structure**.
2 In the dialog box that opens enter `fem_asym_cases` in the **Variable name for FEM structure** edit field.
3 Click **OK**. COMSOL Script opens automatically.

COMBINING THE LOAD CASES

To obtain the total solution you need to superpose or add the solutions from the pressure load with symmetric load cases and the antisymmetric load cases. With the two FEM structures exported to COMSOL Script, add the solution vectors according to Table 3-3 on page 110 and create a new FEM structure, which you can use for the fatigue analysis and import into the graphical user interface to visualize the results. To do this, use the script `wheel_rim_fatigue_superpose.m`, which you find in the `fatigue` directory located in the COMSOL installation directory.

1 With both FEM structures exported to COMSOL Script, run the script `wheel_rim_fatigue_superpose.m` by typing `wheel_rim_fatigue_superpose` at the prompt and pressing Return.

The script `wheel_rim_fatigue_superpose.m` is shown below.

```
% Superpose symmetric and antisymmetric results for the
% wheel rim model

% Export symmetric solution as "fem_sym_cases" and antisymmetric
% solution as "fem_asym_cases"

% Extract solution
us = fem_sym_cases.sol.u;
ua = fem_asym_cases.sol.u;

% Find number of load positions (num_comb)
num_sym_sol = size(us,2);
num_comb = (num_sym_sol - 2) * 2;

% Create the linear combinations
u_tire_pres = us(:,num_sym_sol);
ucomb = zeros(size(us,1), num_comb);
ucomb(:,1) = us(:,1) + u_tire_pres;
ucomb(:,num_sym_sol-1) = us(:,6) + u_tire_pres;
for i = 2:num_sym_sol - 2
  ucomb(:,i) = 0.5 * (us(:,i) + ua(:,i-1)) + u_tire_pres;
```

```
  ucomb(:,num_comb-i+2) = 0.5 * (us(:,i) - ua(:,i-1)) + u_tire_pres;
end

% Create mnemonic parameter values for the combinations
angle_step = 360 / num_comb;
param_list = 0:angle_step:360-angle_step;

% Create new solution object and an fem structure to place it in
sol = femsol(ucomb,'pname','lcmb','plist',param_list);
fem_added = fem_sym_cases;
fem_added.sol = sol;

%fem_added can now be imported into COMSOL Multiphysics
```

Fatigue Analysis

1 To perform the fatigue analysis run the script `wheel_rim_fatigue_analysis.m` by typing `wheel_rim_fatigue_analysis` at the prompt and pressing Return.

The script `wheel_rim_fatigue_analysis.m` is shown below.

```
% Script for running the wheel rim fatigue analysis
[sxx, syy, szz, sxy, syz, sxz] =
posteval(fem,'sx_smsld','sy_smsld', 'sz_smsld', ...

'sxy_smsld','syz_smsld','sxz_smsld','edim',2,'solnum','all','cont
','on','refine',1);
sigma = zeros(6,size(sxx.d,2),size(sxx.d,1));
sigma(1,:,:)=sxx.d';
sigma(2,:,:)=syy.d';
sigma(3,:,:)=szz.d';
sigma(4,:,:)=sxy.d';
sigma(5,:,:)=syz.d';
sigma(6,:,:)=sxz.d';

k = 0.3;
f = 76E6;
params.f = f;
params.k = k;
angle_step = 9;

[resval sigma_max, delta_tau, worst_ind, s_history] = ...
  hcfmultiax(sigma, 'anglestep', angle_step, 'params', params);

findley_res = sxx;
findley_res.d(1,:) = resval;
figure(1)
postdataplot(findley_res, 'colorbar', 'on', ...
    'campos', [0.342,0.923,-2.87], ...
```

```
    'camtarget', [0.0589,0.00860,-0.00152], ...
    'camup', [0.774,0.577,0.260], ...
    'camva', 9.1)

sigma_max_res = sxx;
sigma_max_res.d(1,:) = sigma_max;
figure(2)
postdataplot(sigma_max_res, 'colorbar', 'on')

delta_tau_res = sxx;
delta_tau_res.d(1,:) = delta_tau;
figure(3)
postdataplot(delta_tau_res, 'colorbar', 'on')

x = 1:(size(s_history,2));
figure(4)
plot(x,s_history(1,:),'-',x,s_history(2,:),':',x,s_history(3,:),'
-.',...
  x,s_history(4,:),'--',x,s_history(5,:),'-',x,s_history(6,:),':')
legend('sxx','syy','szz','sxy','syz','sxz')
title('Stress history in critical point')
```

Postprocessing and Visualization

The fatigue analysis script creates plots showing the results of this analysis. In addition, you can plot the stress distribution from the static analysis through the following steps:

1 Switch back to COMSOL Multiphysics and select **File>Import>FEM Structure**.

2 In the **Enter name of FEM structure variable** edit field, enter `fem_added`.

3 Click **OK**.

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

5 Select the **General** page.

6 Under the **Plot type** group label clear the **Slice** check box.

7 Select the **Subdomain** check box.

8 In the **Parameter value** list select **144** to display the von Mises stress for the point of contact at 144°.

9 Select the **Make rough plots** check box.

   The above feature generates plots with slightly lower quality than normal for faster and more memory-efficient rendering.

10 Click **OK**.

10

Fluid-Structure Interaction

This chapter contains models including the interaction between fluids and structures, often named *fluid-structure interaction* (FSI).

Freezing Soil

Introduction

When wet soil or clay is subjected to freezing temperatures, water in the interstices freezes. Because water expands when it freezes, the surrounding soil deforms. The deformation changes the pressure in the interstices. The combined impacts of the freezing and the deformation affect the water flow.

This model predicts 3-way interactions between stress and strain, fluid flow, and temperature change. This type of analysis is important in assessments related to road and building construction, freeze-thaw weathering, fluid flow, and a number of environmental applications. The model that follows comes from a COMSOL client who used the results to assess thermo-mechanical impacts in a transportation study.

This analysis couples equations that predict what happens when a water-filled soil core freezes from the center outwards. Included in the analysis are:

- Effects of a stepwise change in the thermomechanical properties at the phase transition temperature
- Porous fluid-flow behavior involving a temperature-driven contribution
- Stress-strain behavior including loads from thermal expansion and fluid flow
- Heat conduction including phase change and the latent heat of freezing
- Coupling effects between the above-mentioned phenomena.

This discussion assumes temperature changes are fully transient with a quasi-static interaction with fluid flow and solid deformation. The model uses the Darcy's Law and the Convection and Conduction application modes from the Earth Science Module. It also employs a Stress-Strain application mode from the Structural Mechanics Module, an application mode that automates temperature-deformation coupling.

Model Definition[1]

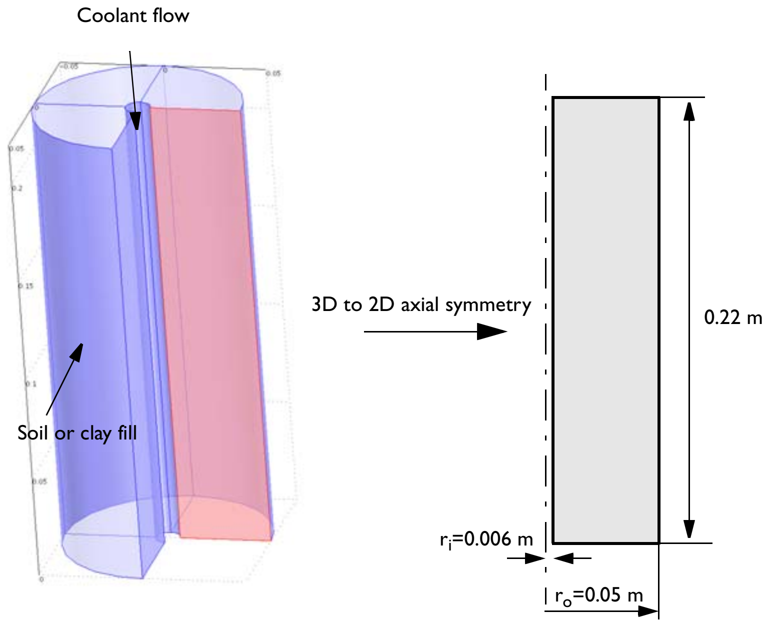


Figure 10-1: Experimental setup for investigations of freezing soil. A symmetry observation permits modeling in axisymmetry 2D.

The model geometry is based on a general test rig often used for investigating the properties of soil and clay (see Figure 10-1). A cylindrical container with the sample soil has a concentric channel, through which a coolant fluid flows. The initial temperature of the wet soil is +3 °C. As the coolant at −15 °C enters the pipe, a freezing front travels outward in the soil specimen. Because the soil is homogeneous, you can take advantage of the geometric symmetry and model the phenomena in 2D.

STRESS-STRAIN EQUATIONS

The fundamental Navier's equation describes a force equilibrium

$$-\nabla \cdot \sigma = \mathbf{F} \tag{10-1}$$

where σ is the stress tensor and $\mathbf{F}$ is a volume force.

The entries of the stress tensor for axisymmetry are

1. Model definition and material data courtesy of Dr. J.P.B.N. Derks, Ministry of Transport, Public Work & Water Management, the Netherlands.

$$\sigma = \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{yz} \end{bmatrix}$$

where τ denotes off-diagonal components of strain or shear.

The equations simplify to two force balances in the r and z directions:

$$\frac{\partial \sigma_r}{\partial r} + \frac{\partial \tau_r}{\partial r} + \frac{\sigma_r - \sigma_\theta}{r} + F_r = 0 \tag{10-2}$$

$$\frac{\partial \tau_{rz}}{\partial r} + \frac{\partial \sigma_z}{\partial r} + \frac{\tau_{rz}}{r} + F_z = 0 \tag{10-3}$$

where τ denotes off-diagonal components of strain or shear.

Flow-to-Structure Coupling

When considering the impacts of fluid flow on structural deformation, the stress tensor decompose into two parts

$$\sigma = \sigma' + \mathbf{m}p \tag{10-4}$$

where σ is the total stress, σ' is the so-called grain stress, and p is the pressure of the fluid moving through a porous sand or clay matrix.

In this analysis

$$\mathbf{m} = \begin{bmatrix} 1 & 1 & 1 & 0 & 0 & 0 \end{bmatrix}^T .$$

Temperature-to-Structure Coupling

The thermomechanical relationship is given by the generalized Hooke's law for an elastic nonisothermal material as in

$$\sigma' = \mathbf{D}(\varepsilon - \varepsilon_{\mathrm{th}}) , \qquad \varepsilon_{\mathrm{th}} = \alpha(T - T_{\mathrm{ref}}) . \tag{10-5}$$

Here $\mathbf{D}$ is the elasticity matrix, σ' represents the elastic stress, ε gives the total strain, and $\varepsilon_{\mathrm{th}}$ is the thermal strain. Further, α (K^{-1}) is the coefficient of thermal expansion, T is the temperature, and T_{ref} is the strain reference temperature.

FLUID FLOW EQUATIONS

Model the flow with the modified Darcy's law

$$\mathbf{u} = -\frac{\kappa}{\eta}\nabla(p + \phi_s) \tag{10-6}$$

where $\mathbf{u} = (u, v, w)$ denotes the vector of fluid velocities in the x, y, and z directions, and ϕ_s is the suction pressure.

Temperature-to-Flow Coupling, Segregation Potential

The fluid flow and temperature relationships couple through the term

$$\phi_s = SP_0 \cdot T/\kappa,$$

where SP_0 is the *segregation potential* (kg·m/(s^2·K)), which is the ratio of the moisture migration velocity to the temperature gradient in a freezing soil, and κ is the permeability (m^2). The segregation potential SP_0 is a positive constant below the freezing point and 0 above. Experimental observations on specimens frozen under a temperature gradient suggest that, even though much of the pore water is frozen, water transport still occurs in the frozen soil past the pore freezing front in response to temperature-induced unfrozen water content gradients and suction gradients in these unfrozen water films. The migratory water freezes at the segregation freezing temperature, T_s, which is lower than the pore freezing temperature T_p (Ref. 1). In this model example, it is assumed that the segregation freezing temperature is well below the temperature range of the study.

Given the definition for ϕ_s, Equation 10-6 states that the fluid velocities depend on the pressure gradient and the temperature gradient for conditions below the freezing point.

Structure-to-Flow Coupling

For quasi-steady flow, the following relationship holds:

$$\nabla \cdot \mathbf{u} = -(\dot{\varepsilon}_{xx} + \dot{\varepsilon}_{yy} + \dot{\varepsilon}_{zz}), \tag{10-7}$$

where $\dot{\varepsilon}_{xx}$ and similar terms are the *rates of strain* (s^{-1}) from the stress-strain equations.

Combining Equation 10-6 and Equation 10-7 gives the governing equation

$$\nabla \cdot \left(-\frac{k}{\eta}\nabla(p + \phi_s)\right) = -(\dot{\varepsilon}_{xx} + \dot{\varepsilon}_{yy} + \dot{\varepsilon}_{zz}), \tag{10-8}$$

which this example models with the Darcy's Law application mode.

TEMPERATURE EQUATIONS

This problem uses the well-known heat equation to model the transfer of heat. As described in the *Earth Science Module User's Guide*, the heat transfer equation reads

$$\rho c_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T) = Q \,.$$

Results

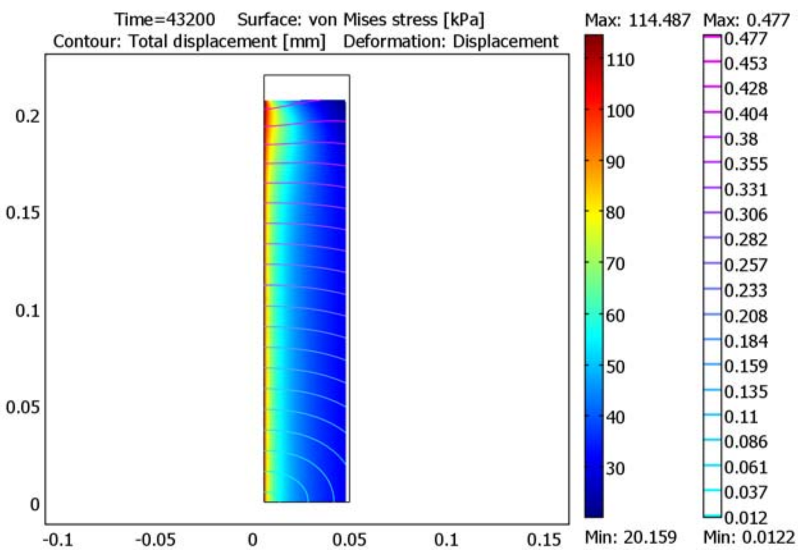


Figure 10-2: A snapshot of the von Mises stresses (surface plot) and displacements (contours) in a column of freezing soil.

Figure 10-2 shows the displacements in the solid sample and the von Mises stresses after 12 hours of freezing operation. It is also easy to monitor how the physical properties of the sand change with time and space; see Figure 10-3.

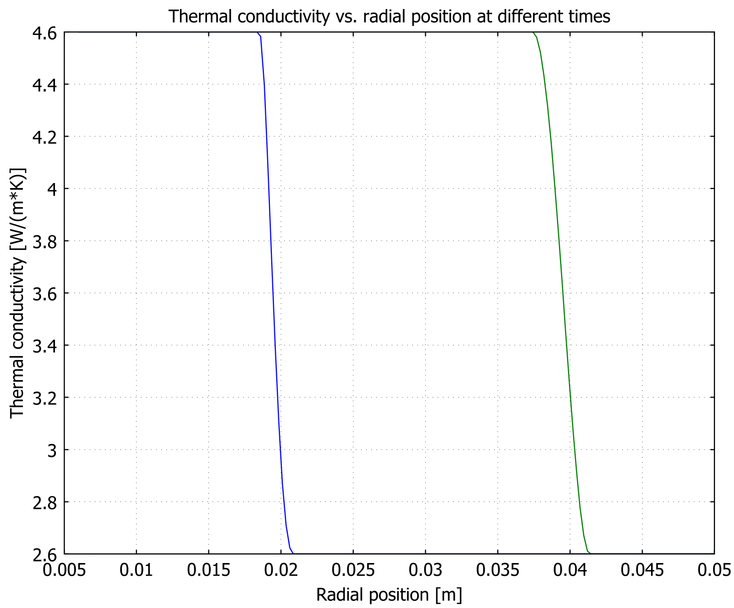


Figure 10-3: Thermal conductivity changes in a step at the freezing point. The lower curve corresponds to 24 minutes, and the upper curve to 7 hours and 12 minutes.

Reference

1. Jean-Marie Konrad, "Estimation of the segregation potential of fine-grained soils using the frost heave response of two reference soils," *Can. Geotech. J.*, vol. 42, pp. 38–50, 2005.

Modeling in COMSOL Multiphysics

Turning to the COMSOL Multiphysics Structural Mechanics Module, you choose the Axial Symmetry, Stress-Strain application mode to solve Equation 10-2 and Equation 10-3. To account for the fluid pressure according to Equation 10-4, simply add

$$\begin{bmatrix} -\frac{\partial p}{\partial r} \\ -\frac{\partial p}{\partial z} \end{bmatrix}$$

to the body load vector $\mathbf{F}$ on the **Load** page in the **Subdomain Settings** dialog box. To have that application mode automatically account for the thermomechanical relations, Equation 10-5, select the **Include thermal expansion** check box on the **Load** page. To get easy access to the rates of strain (time derivatives of the strain), use a time-dependent stress-strain analysis. This solves Equation 10-1 with an extra term on the left-hand side, namely $\rho(\partial^2\mathbf{u})/(\partial t^2)$, which is the acceleration term in Newton's second law. Here, $\mathbf{u}$ is the vector of directional deformations (in m). However, you can make the assumption that the time scale in the mechanical problem is much shorter than that for the heat transfer problem. Here you skip the time-derivative term in the stress/strain equation by setting $\rho = 0$ and still access the rates of strain as described in the next paragraph.

It is easy to implement the modified Darcy's law (Equation 10-3) with the Earth Science Module's Darcy's Law application mode, and its predefined equation is

$$\nabla \cdot \left(-\frac{\kappa}{\eta}\nabla(p + \rho_f g D)\right) = Q_s . \tag{10-9}$$

Because the vertical change is small, you can ignore gravity impacts on flow. There is, however, another contribution to the mass flux, namely that from the segregation potential (see Equation 10-8). Include this contribution by setting $D = \phi_s/(g\rho_f)$.

Next place the rate-of-strain expression in Equation 10-7 in the Q_s term. The r, ϕ , and z components of the rate of strain, $\dot{\varepsilon}_{rr}$, $\dot{\varepsilon}_{\phi\phi}$ and $\dot{\varepsilon}_{zz}$, are readily available in the time-dependent Axial Symmetry, Stress-Strain application mode as the expressions `diff(er_stress,t)`, `diff(ephi_stress,t)`, and `diff(ez_stress,t)`. Hence, enter `-(diff(er_stress,t)+diff(ephi_stress,t)+diff(ez_stress,t))` in the Q_s edit field of the Darcy's Law application mode.

The model handles the stepwise-changing material properties at the freezing point as well as the latent heat of freezing as described in "Phase Change" on page 303 in the *Earth Science Module Model Library.*

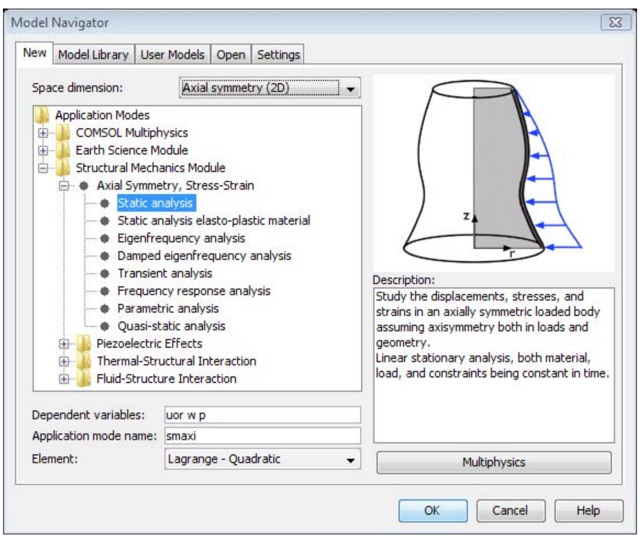
Note: This model requires the Earth Science Module and the Structural Mechanics Module.

Model Library path: `Structural_Mechanics_Module/Fluid-Structure_Interaction/freezing_soil`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator**, select **Axial symmetry (2D)** from the **Space dimension** list and then in the list of application modes select **Structural Mechanics Module>Axial Symmetry, Stress-Strain>Static analysis**. In the **Application mode name** edit field, type `stress`.



2 Click the **Multiphysics** button, then click **Add**.

3 Return to the list of application modes and select **COMSOL Multiphysics>Heat Transfer>Conduction>Transient Analysis**. In the **Application mode name** edit field enter `heat`, then click **Add**.

4 In the list of application modes select **Earth Science Module>Fluid Flow> Darcy's Law>Pressure analysis**. In the **Application mode name** edit field enter `Darcy`, then click **Add**.

5 Click **OK**.

OPTIONS

1 From the **Options** menu select **Constants** and enter the following names, expressions, and descriptions (optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T_trans	0[degC]	Freezing point
scale	0.5	Width of smoothed step function
rho_sa	2000[kg/m^3]	Density
nu0	0.3	Poisson's ratio
alpha_s	0.8e-6[1/K]	Coefficient of thermal expansion
T_init	3[degC]	Intial temperature
k_sf	4.6[W/(m*K)]	Thermal conductivity below freezing point
k_s	2.6[W/(m*K)]	Thermal conductivity above freezing point
Cp_sf	1000[J/(kg*K)]	Heat capacity below freezing point
Cp_s	1350[J/(kg*K)]	Heat capacity above freezing point
kappa_s	7.1e-5[m^2]	Permeability, unfrozen sand
eta_s	0.001[Pa*s]	Dynamic viscosity
E0_s	65[MPa]	Young's modulus
t_load	300[kN/m^2]	Top edge load
e_load	300[kN/m^2]	Side edge load
dT	0.5[K]	Half width of Gauss bell curve
lam	333[kJ/kg]	Latent heat of freezing
p_pore	150[kPa]	Pore pressure

GEOMETRY MODELING

1 Shift-click the **Rectangle/Square** button at the top of the Draw toolbar.

2 In the **Size** area, set the **Width** to `0.044` and the **Height** to `0.22`.

3 In the **Position** area, select **Center** from the **Base** list. Set **r** to `0.028` and **z** to `0.11`.

4 Click **OK**, then click the **Zoom Extents** button on the Main toolbar.

EXPRESSION DEFINITIONS

From the **Options** menu select **Expressions>Scalar Expressions**, then add the following names, expressions, and descriptions (optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
HS	`flc1hs((T-T_trans)[1/K],scale)`	`Smoothed Heaviside step`
k_g	`k_sf+(k_s-k_sf)*HS`	`Varying thermal conductivity`
D	`exp(-(T-T_trans)^2/dT^2)/ sqrt(pi*dT^2)`	`Smoothed Dirac delta function`
Cp	`Cp_sf+(Cp_s-Cp_sf)*HS`	`Varying heat capacity`
T_b	`T_init+18[K]*(1- 2*flc1hs(t[1/s],100))`	`Varying boundary temperature`
Sp0	`(1-HS)*1e-12[kg*m/(s^2*K)]`	`Varying segregation potential`

The unit brackets `[1/K]` and `[1/s]` make the function inputs dimensionless.

PHYSICS SETTINGS

Application Scalar Variables

To include the segregation potential mass-flux term in the way described below Equation 10-9, you need to modify the application scalar variable D_{Darcy}.

1 From the **Physics** menu, select **Scalar Variables**.

2 In the **Expression** column for **D_Darcy**, type `(Sp0*T/kappa_s)/ (g_Darcy*rhof_Darcy)`.

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

Subdomain Settings—Axial Symmetry, Stress-Strain

1 From the **Multiphysics** menu, select **Axial Symmetry, Stress-Strain (stress)**.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select Subdomain 1, then enter the values in the following table:

QUANTITY	VALUE/EXPRESSION	DESCRIPTION
E	`E0_s`	Young's modulus
ν	`nu0`	Poisson's ratio
α	`alpha_s`	Thermal expansion coeff.

4 On the **Load** page, select the **Include thermal expansion** check box.

5 Specify the loads as in the following table; when done, click **OK**.

QUANTITY	VALUE/EXPRESSION	DESCRIPTION
F_r	`-p2r`	Body load (force/volume) r dir.
F_z	`-p2z`	Body load (force/volume) z dir.
Temp	`T`	Strain temperature
Tempref	`T_init`	Strain ref. temperature

Boundary Conditions—Axial Symmetry, Stress-Strain

1. Select the menu **Physics>Boundary Settings**.
2. On the **Constraint** page select Boundaries 1 and 2. From the **Constraint condition** list select **Roller**.
3. Click the **Load** tab. Select Boundary 3, and set $\mathbf{F_r}$ to `0` and $\mathbf{F_z}$ to `-t_load`.
4. Select Boundary 4. Set $\mathbf{F_r}$ to `-e_load` and $\mathbf{F_z}$ to `0`.
5. Click **OK**.

Subdomain Settings—Heat Transfer

1. From the **Multiphysics** menu select **Heat Transfer by Conduction (heat)**.
2. Choose **Physics>Subdomain Settings**, then specify these settings:

QUANTITY	VALUE/EXPRESSION	DESCRIPTION
k (isotropic)	`k_g`	Thermal conductivity
ρ	`rho_s`	Density
C_p	`Cp+D*lam`	Heat capacity

3. On the **Init** page, set $\mathbf{T(t_0)}$ to `T_init`. Click **OK**.

Boundary Conditions—Heat Transfer

1. Choose **Physics>Boundary Settings** and select Boundary 1. From the **Boundary condition** list select **Temperature**, then in the **Temperature** edit field type `T_b`.
2. Select Boundary 4. In the **Boundary condition** list select **Temperature**, then in the **Temperature** edit field type `T_init`. Click **OK**.

Subdomain Settings—Darcy's Law

1. Select the menu item **Multiphysics>Darcy's Law (Darcy)**.
2. From the **Physics** menu, select **Subdomain Settings**.

3 Specify the following settings; when done, click **OK**.

QUANTITY	VALUE/EXPRESSION	DESCRIPTION
κ_s	`kappa_s`	Saturated permeability
ρ_f	`1`	Density, liquid
η	`eta_s`	Viscosity, liquid
Q_s	`-(diff(er_stress,t)+diff(ephi_stress,t) +diff(ez_stress,t))`	Liquid source

Because of the setting for D on page 413, the density value has to be nonzero but is otherwise irrelevant.

Boundary Conditions—Darcy's Law

Choose **Physics>Boundary Settings**, then select Boundary 1. In the **Boundary condition** list select **Pressure**, then set $\mathbf{p_0}$ to `p_pore`. Similarly select Boundary 4 and set the **Pressure** to `0`. Click **OK**.

MESH GENERATION

1. From the **Mesh** menu, select **Free Mesh Parameters**.
2. Click the **Boundary** tab and select Boundary 1. In the **Maximum element size** edit field enter `0.003`.
3. Click **Remesh**. When the mesher has finished, click **OK**.

COMPUTING THE SOLUTION

1. Click the **Solver Parameters** button on the Main toolbar.
2. In the **Times** edit field, type `0:1440:43200`. This provides 30 equidistant time steps during a total simulation time of 12 hours (43,200 s).
3. In the **Absolute tolerance** edit field, type `uor 1e-6 w 1e-6 T 0.001 p2 10`. This sets up absolute tolerances of 10^{-6} for the displacement variables, 0.001 for the temperature, and 10 for the pressure. Click **OK**.
4. Click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

To reproduce the plot in Figure 10-2, follow these steps.

1. Click the **Plot Parameters** button on the Main toolbar.
2. On the **General** page, go to the **Plot type** area and select the **Surface**, **Contour**, **Deformed shape**, and **Geometry edges** check boxes.

3 Click the **Surface** tab, then in the **Predefined quantities** list on the **Surface Data** page select **Axial Symmetry, Stress-Strain (stress)>von Mises Stress**.

4 From the **Unit** list, select **kPa**.

5 Click the **Contour** tab, and in the **Predefined quantities** list on the **Contour Data** page select **Axial Symmetry, Stress-Strain (stress)>Total displacement**.

6 From the **Unit** list, select **mm**.

7 Click **OK** to close the **Plot Parameters** dialog box and generate the plot.

Obstacle in Fluid

Introduction

In this model, fluid is flowing through a channel with a flexible obstacle. Due to the viscous and pressure forces exerted by the fluid, the obstacle is bending. With the obstacle undergoing a large deformation, the fluid flow domain is also changing considerably. COMSOL Multiphysics is taking these changes into account by computing the flow field on a moving mesh attached to the obstacle.

In this example model, the dimensions are such that the setup mostly resembles a MEMS flowmeter, but the modeling principles used are very general and can be applied to many situations where there is interaction between a structure and a fluid flow domain.

Model Definition

In this example the flow channel is 200 µm long, 150 µm high, and 150 µm wide. A flag-shaped obstacle has a slightly off-center position in the channel. The fluid is water with a density $\rho = 1000\ \mathrm{kg/m^3}$ and dynamic viscosity $\eta = 0.001\ \mathrm{Pa{\cdot}s}$. The obstacle is made of a flexible material with a density $\rho = 7850\ \mathrm{kg/m^3}$ and a Young's modulus $E = 8\ \mathrm{MPa}$.

The model consists of a fluid part, solved with the Navier-Stokes equations in the flow channel, and a structural mechanics part, which you solve in the obstacle. A Moving Mesh application mode makes sure the flow domain is deformed along with the obstacle. These application modes and the FSI-specific settings on subdomains and boundaries are available directly when using the predefined multiphysics couplings for fluid-structure interaction.

FLUID FLOW

The fluid flow in the channel is described by the Navier-Stokes equations, solving for the velocity field $\mathbf{u} = (u, v)$ and the pressure, p:

$$
\begin{aligned}
&\rho\frac{\partial\mathbf{u}}{\partial t} - \nabla\cdot[-p\mathbf{I} + \eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)] + \rho(\mathbf{u}\cdot\nabla)\mathbf{u} = \mathbf{F} \\
&-\nabla\cdot\mathbf{u} = 0
\end{aligned}
$$

where $\mathbf{I}$ is the unit diagonal matrix, and $\mathbf{F}$ is the volume force affecting the fluid. We assume no gravitation or other volume forces affecting the fluid, so $\mathbf{F} = 0$.

The Navier-Stokes equations are solved in the spatial (deformed) coordinate system. At the inlet, the model uses a fully developed laminar flow. Zero pressure is applied at the outlet. No-slip boundary conditions, that is $\mathbf{u} = 0$, are used at all other boundaries. Note that this is a valid condition only as long as you are solving the stationary problem. In a transient version of the same model, with the obstacle starting out from an undeformed state, it is necessary to state that the fluid flow velocity be the same as the velocity of the deforming obstacle. This is done in “ALE Fluid-Structure Interaction” on page 294 in the *MEMS Module Model Library*, also using the predefined FSI interface, where these velocities are available directly.

STRUCTURAL MECHANICS

The structural deformations are solved for using an elastic formulation and a nonlinear geometry formulation to allow large deformations.

For boundary conditions, the obstacle is fixed to the bottom of the fluid channel so that it cannot move in any direction. All other boundaries experience a load from the fluid, given by

$$\mathbf{F}_{\mathrm{T}} = -\mathbf{n} \cdot (-p\mathbf{I} + \eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^{T})) \tag{10-10}$$

where $\mathbf{n}$ is the normal vector to the boundary. This load represents a sum of pressure and viscous forces. In addition, the predefined fluid load takes the area effect between the reference frame for the solid and the moving ALE frame in the fluid into account.

MOVING MESH

The motion of the deformed mesh is modeled using Winslow smoothing. For more information on this formulation, please refer to “The Moving Mesh Application Mode” on page 401 in the *COMSOL Multiphysics Modeling Guide*. The boundary conditions control the displacement of the moving mesh with respect to the initial geometry. At the boundaries of the obstacle, this displacement is the same as the structural deformation. At the exterior boundaries of the flow domain, it is set to zero in all directions. These settings are available as predefined groups.

Results

Figure 10-4 shows the velocity field and the deformed obstacle for a maximum inlet velocity of 1.5 m/s. With deformations of this magnitude, the changes in the fluid flow domain have a visible effect on the flow.

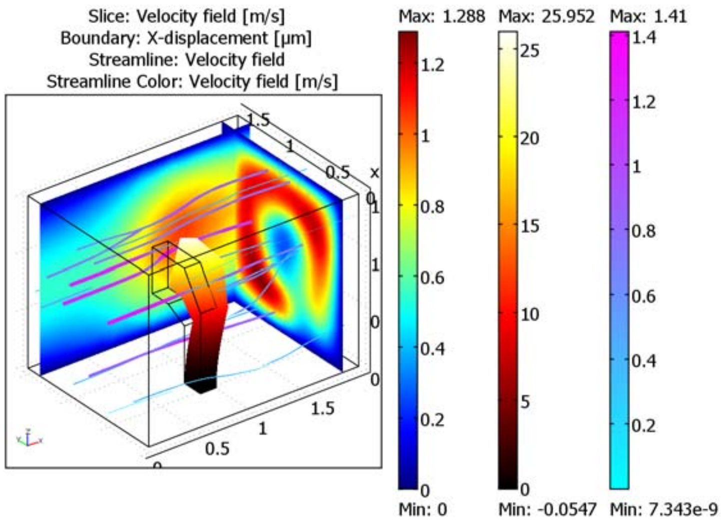


Figure 10-4: Flow velocity (m/s) in two cross sections of the flow channel and the deformation (μm) of the obstacle. The color and direction of the streamlines indicate the velocity and the direction of the flow.

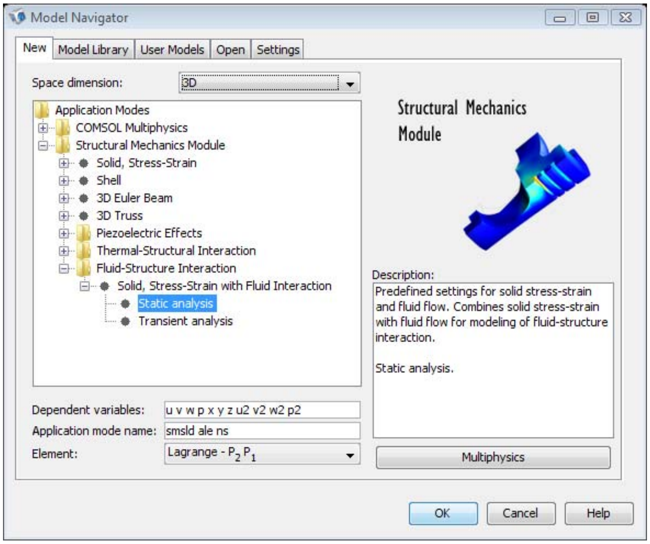
Modeling Using the Graphical User Interface

Model Library path: `Structural_Mechanics_Module/Fluid-Structure_Interaction/obstacle_in_fluid`

MODEL NAVIGATOR

1 In the **Model Navigator**, select **3D** from the **Space dimension** list and click the **Multiphysics** button.

2 From the list of application modes, select **Structural Mechanics Module> Fluid-Structure Interaction>Solid, Stress-Strain with Fluid Interaction>Static analysis**.



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

GEOMETRY MODELING

1 Draw a block with **Length X**: `2e-4`, **Y**: `1.5e-4`, **Z**: `1.5e-4`, and the **Axis base point** in the origin.

2 Click the **Zoom Extents** button to see the block.

3 Go to **Draw>Work-Plane Settings**. In the dialog box that appears, select a **y-z** plane placed at **x** = `0.8e-4`. Click **OK** to close the dialog box. This takes you to the work plane.

4 To see the block, click on the **Projection of 3D Geometries** toolbar button on the Visualization/Selection toolbar to the left of the drawing area.

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

6 Go to **Options>Axes/Grid Settings**. In the dialog box that appears, apply the settings in the table below. To set the grid spacing you need to go to the **Grid** page and clear the **Auto** check box.

SETTING	VALUE
Grid x spacing	`1e-5`
Grid y spacing	`1e-5`

7 Click the **Line** button and click on each coordinate in the table below, working your way from Point 1 to Point 8. When done, right-click to complete the curve. Your geometry should now look like Figure 10-5.

X COORD	Y COORD
`0.4e-4`	`0`
`0.4e-4`	`0.7e-4`
`0.6e-4`	`1.0e-4`
`1.0e-4`	`1.0e-4`
`1.0e-4`	`0.6e-4`
`0.8e-4`	`0.6e-4`
`0.6e-4`	`0.5e-4`
`0.6e-4`	`0`

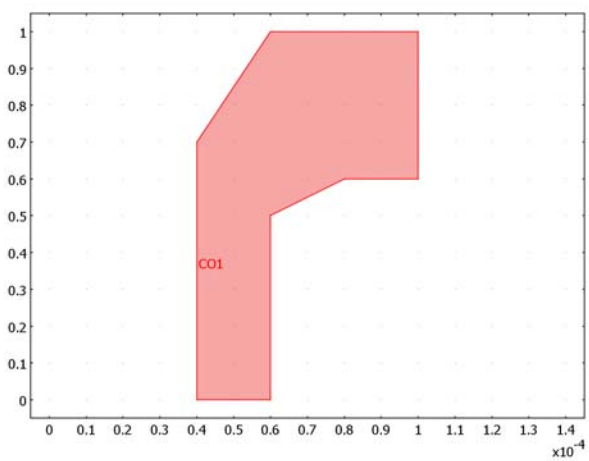


Figure 10-5: Cross section of the obstacle.

8 Go to **Draw>Extrude** and extrude your object by the distance `0.15e-4`. Leave all other settings at their default values and click **OK**.

PHYSICS SETTINGS

1 Go to **Options>Constants** and define the following constants. The descriptions are optional..

NAME	EXPRESSION	DESCRIPTION
`u_max`	`1.5[m/s]`	Maximum inlet velocity
`width`	`150[um]`	Channel width
`height`	`150[um]`	Channel height

Subdomain Settings

1 From the **Multiphysics** menu, make sure **3 Geom1: Incompressible Navier-Stokes (ns)** is selected.

2 In the **Subdomain Settings** dialog box, select Subdomain 1 and select **Fluid domain** from the **Group** list beneath the **Subdomain** list. Then apply the following settings:

SUBDOMAIN	1
ρ	`1e3`
η	`1e-3`

3 Click the **Artificial Diffusion** button.

4 In the **Artificial Diffusion** dialog box, clear the **Streamline diffusion** check box and then click **OK**.

5 Select Subdomain 2 and select **Solid domain** from the **Group** list beneath the **Subdomain** list.

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

7 From the **Multiphysics** menu, select **1 Geom1: Solid, Stress-Strain (smsld)**.

8 In the **Subdomain Settings** dialog box, select Subdomain 1 and then select **Fluid domain** from the **Group** list.

9 Select Subdomain 2 and select **Solid domain** from the **Group** list. Then enter `8e6` in the **E** edit field for Young's modulus.

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

11 From the **Multiphysics** menu, select **2 Geom1: Moving Mesh (ALE) (ale)**.

12 In the **Subdomain Settings** dialog box, apply the following settings by selecting from the **Group** list; when done, click **O**K.

SETTING	SUBDOMAIN 1	SUBDOMAIN 2
Group	Fluid domain	Solid domain

Boundary Conditions

1 From the **Multiphysics** menu, select **Incompressible Navier-Stokes**.

2 In the **Boundary Settings** dialog box, apply the following boundary conditions on the active boundaries:

SETTINGS	BOUNDARY 1	BOUNDARY 16	ALL OTHER BOUNDARIES
Boundary type	Inlet	Outlet	Wall
Boundary condition	Velocity	Pressure, no viscous stress	No slip
u_0	`16*u_max*y*(width-y)*z*(height-z)/(width^2*height^2)`		
v_0	0		
w_0	0		
p_0		0	

3 Click **OK**.

4 From the **Multiphysics** menu, select **Solid, Stress-Strain**.

5 In the **Boundary Settings** dialog box, apply the following boundary settings on the active boundaries:

SETTINGS	BOUNDARY 8		ALL OTHER BOUNDARIES	
Page	Constraints			
	Constraint condition	Fixed	Group	Fluid load

6 On the boundaries that are not fixed, select **Fluid load** from the **Group** list to define the fluid load on the solid domain.

7 From the **Multiphysics** menu, select **Moving Mesh (ALE)**.

8 In the **Boundary Settings** dialog box, apply the following predefined groups of settings on the active boundaries:

SETTINGS	BOUNDARIES 1–5, 16	ALL OTHER BOUNDARIES
Group	Fixed	Structural displacement

MESH GENERATION

1 Open the **Free Mesh Parameters** dialog box and set **Predefined mesh sizes** to **Coarse**.

2 On the **Subdomain** page, select Subdomain 2 and enter `0.2e-4` in the **Maximum element size** edit field.

3 Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 On the **General** page of the **Plot Parameters** dialog box, select the check boxes for **Slice**, **Boundary**, **Streamline**, and **Geometry edges**.

2 On the **Boundary** page, select **Solid, Stress-Strain (smsld)>X-displacement** from the **Predefined quantities** list. Select **μm** from the **Unit** list. Select **hot** from the **Colormap** list.

3 On the **Streamline** page, select **Incompressible Navier-Stokes (ns)>Velocity field** from the **Predefined quantities** list. Set the **Line type** to **Tube**. Click the **Tube Radius** button and set the **Radius scale factor** to `0.5`. On the **Line Color** page, select **Use expression**. Click the **Color Expression** button, then select **Incompressible Navier-Stokes (ns)> Velocity field** from the **Predefined quantities** list. Set the **Colormap** to **cool**, then click **OK**.

4 On the **Slice** page, select **Incompressible Navier-Stokes (ns)>Velocity field** from the **Predefined quantities** list. Use the **Vector with coordinates** edit field to put one slice at **x** = `1.85e-4` and another one at **y** = `1.35e-4`.

5 Click **OK** to generate the plot.

11

Fracture Models

This section contains examples of fracture models that you can study using the Structural Mechanics Module.

Single Edge Crack

Introduction

This model deals with the stability of a plate with an edge crack that is subjected to a tensile load. To analyze the stability of exciting cracks, you can use fracture mechanics.

A common parameter in fracture mechanics, the so-called stress intensity factor K_I, provides a means to predict if a specific crack will cause the plate to fracture. When this calculated value becomes equal to the critical fracture toughness of the material K_{Ic} (a material property), then fast, usually catastrophic fracture occurs.

THE J-INTEGRAL

In this model, you determine the stress intensity factor K_I using the so-called J-integral.

The J-integral is a two-dimensional line integral along a counterclockwise contour, Γ, surrounding the crack tip. The J-integral is defined as

$$J = \int_\Gamma W dy - T_i \frac{\partial u_i}{\partial x} ds = \int_\Gamma \left(W n_x - T_i \frac{\partial u_i}{\partial x} \right) ds$$

where W is the strain energy density

$$W = \frac{1}{2}(\sigma_x \cdot \varepsilon_x + \sigma_y \cdot \varepsilon_y + \sigma_{xy} \cdot 2 \cdot \varepsilon_{xy})$$

and $\mathbf{T}$ is the traction vector defined as

$$\mathbf{T} = [\sigma_x \cdot n_x + \sigma_{xy} \cdot n_y, \sigma_{xy} \cdot n_x + \sigma_y \cdot n_y] .$$

σ_{ij} denotes the stress components, ε_{ij} the strain components, and n_i the normal vector components.

The J-integral has the following relation to the stress intensity factor for a plane stress case and a linear elastic material:

$$J = \frac{K_I^2}{E}$$

where E is Young's modulus.

ANALYTICAL SOLUTION

This problem has an analytical solution (see Ref. 1) for the stress intensity factor:

$$K_{\mathrm{Ia}} = \sigma \cdot \sqrt{\pi \cdot a} \cdot \mathrm{ccf}$$

where $\sigma = 20$ MPa (tensile load (force /area)), $a = 0.6$ m (crack length), and $\mathrm{ccf} = 2.1$ (configuration correction factor). This correction factor is determined for this configuration using an polynomial equation from Ref. 1. This gives the following analytical stress intensity factor: $K_{\mathrm{Ia}} = 57.66$ MPa·m$^{1/2}$.

Model Definition

GEOMETRY

A plate with a width w of 1.5 m has a single horizontal edge-crack of length $a = 0.6$ m on the left vertical edge (see Figure 11-1). The total height of the plate is 3 m, but due to symmetry reasons the model only includes half of the plate. The model geometry also includes two smaller domains, thus adding interior boundaries.

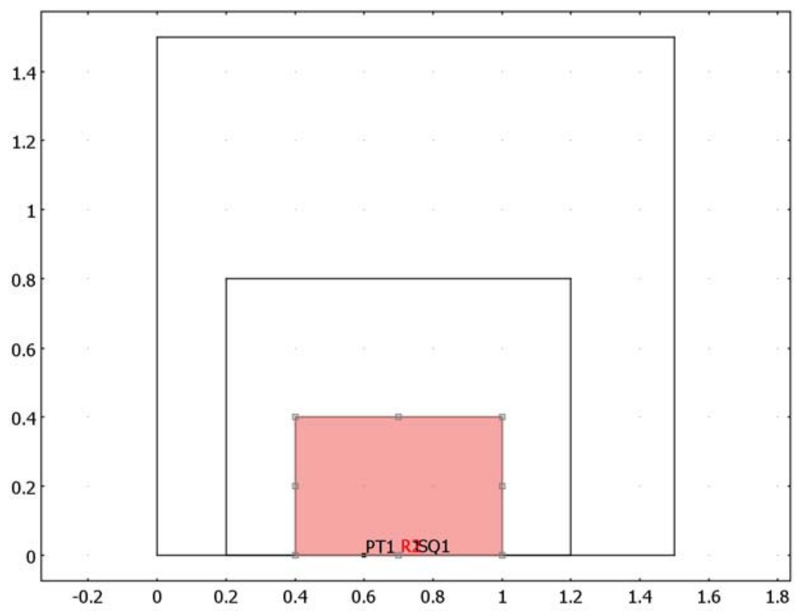


Figure 11-1: Plate geometry.

DOMAIN EQUATIONS

Due to the interior boundaries the geometry consists of three subdomains. The same material properties apply to all three domains:

QUANTITY	NAME	EXPRESSION
Young's modulus	E	$206 \cdot 10^9$
Poisson's ratio	ν	0.3

BOUNDARY CONDITIONS

You apply a tensile load to the upper horizontal edge, while the lower horizontal edge is constrained in the y direction from $x = 0.6$ m to $x = 1.5$ m.

Modeling in COMSOL Multiphysics

The analysis computes the J-integrals for three different contours traversing three different regions around the crack tip. The first contour follows the exterior boundaries of the plate. The second contour follows the interior boundaries at x = 0.2 m, y = 0.8 m, and x = 1.2 m. The third and last contour follows the interior boundaries at x = 0.4 m, y = 0.4 m, and x = 1 m.

To calculate the J-integral, you define boundary integration variables for each contour. This creates two variables for each contour. The first variable, denoted W, contains the integrated strain energy density, while the second, denoted **Tdudx**, contains the traction vector times the spatial x-derivative of the deformation components. The sum of these two variables then provides the J-integral value as a scalar expression variable. Finally, you can compute the stress intensity factor from the J-integral value, also using a scalar expression variable.

Note that the boundaries along the crack are not included in the J-integral, because they do not give any contribution to the J-integral. This is due to the following facts: for an ideal crack dy is zero along the crack faces, and all traction components are also zero ($T_i = 0$) as the crack faces are not loaded.

When calculating the J-integral, the contour normals must point outward of the region which the contour encloses. The boundary normals of the plate geometry are all pointing outward. However, some of the normals of the interior boundaries that are used for the second and third contour do not point outward of the regions that they enclose. This means that you need to define additional normals to get the correct alignment of the normals. You define these normals as boundary expressions and use them in the boundary integration variables.

Results

The following table shows the stress intensity factors for the three different contours:

CONTOUR	STRESS INTENSITY FACTOR
1	57.79 $MPa{\cdot}m^{1/2}$
2	57.71 $MPa{\cdot}m^{1/2}$
3	57.67 $MPa{\cdot}m^{1/2}$

It is clear from these results that the values for the stress intensity factor in the COMSOL Multiphysics model are in good agreement with the reference value for all contours.

It is also clear that the accuracy of the calculated stress intensity factors increases for the inner contours.

Figure 11-2 shows the stress singularity at the crack tip.

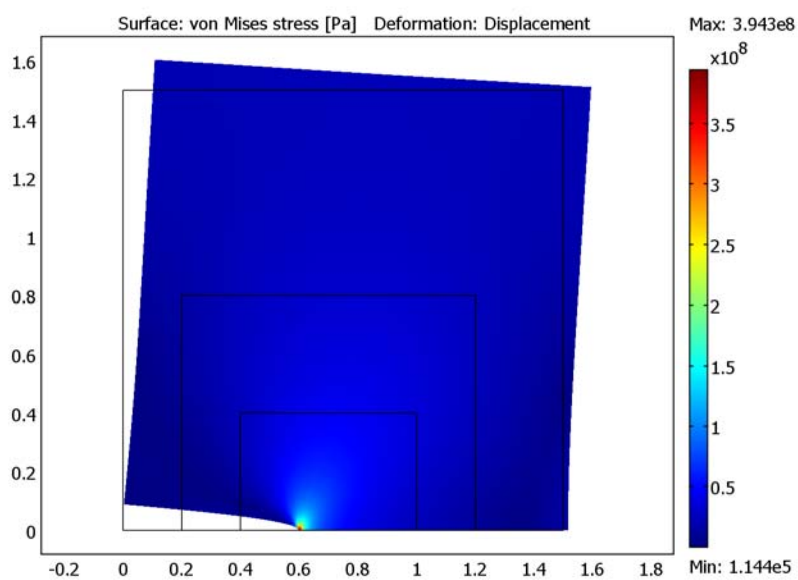


Figure 11-2: Von Mises stresses and the deformed shape of the plate.

Reference

1. Abdel-Rahman Ragab and Salah Eldin Bayoumi, *Engineering solid mechanics*. CRC Press, 1998.

Model Library path: `Structural_Mechanics_Module/Fracture/single_edge_crack`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Select **2D** in the **Space dimension** list.
2. Select **Structural Mechanics Module>Plane Stress** in the list of application modes.
3. Click **OK**.

GEOMETRY MODELING

First create the upper half of the plate and add two rectangles so that the inner boundaries which is created can be used as contours.

1. Choose **Draw>Specify Objects>Square**.
2. In the **Width** edit field type `1.5`.
3. Click **OK**.
4. Add a rectangle by choosing **Draw>Specify Objects>Rectangle**.
5. In the **Width** edit field type `1` and in the **Height** edit field type `0.8`.
6. In the **x** edit field type `0.2` and in the **y** edit field type `0`.
7. Click **OK**.
8. To add a second rectangle, first choose **Draw>Specify Objects>Rectangle**.
9. In the **Width** edit field type `0.6` and in the **Height** edit field type `0.4`.
10. In the **x** edit field type `0.4` and in the **y** edit field type `0`.
11. Click **OK**.

    Next add a point to define the crack tip.

12. Choose **Draw>Specify Objects>Point**.
13. In the **x** edit field type `0.6` and in the **y** edit field type `0`.

14 Click **OK**.

This completes the geometry. Compare your result with Figure 11-1 on page 427.

PHYSICS SETTINGS

Subdomain Settings

1. From the **Physics** menu, choose **Subdomain Settings** edit field.
2. Select Subdomains 1, 2, and 3.
3. In the **Young's modulus** edit field type `2e11`.
4. In the **Poisson's ratio** edit field type `0.3`.
5. Click **OK**.

Boundary Conditions

1. From the **Physics** menu choose **Boundary Settings**.
2. Select Boundaries 10, 12, and 14, then select **Symmetry plane** from the **Constraint condition** list.
3. Click the **Load** tab.
4. Select Boundary 3. In the $\mathbf{F_y}$ edit field type `20e6`.
5. Click the **Edge load is defined as force/area using the thickness** option button, then click **OK**.

Point Conditions

1. From the **Physics** menu choose **Point Settings**.
2. Select Point 7 and select the $\mathbf{R_x}$ check box. Leave the displacement at `0`.
3. Click **OK**.

Boundary Expressions

The interior boundaries, which you use for the two inner J-integral contours, need to have user-defined normal direction variables.

1. Choose **Options>Expressions>Boundary Expressions**.

2 Specify expressions for each of the normal vector components according to the following table. When finished, click **OK**.

BOUNDARY	NAME	EXPRESSION
4, 7	Nx Ny	-1 0
6, 9	Nx Ny	0 1
11, 13	Nx Ny	1 0

Boundary Integration Variables

1 From the **Options** menu, choose **Integration Coupling variables** and click on **Boundary Variables**.

2 Select Boundaries 1, 3, and 15.

3 Specify expressions according to the following table:

NAME	EXPRESSION
W	(sx_smps*ex_smps+sy_smps*ey_smps+2*sxy_smps*exy_smps)*nx/2
Tdudx	-((sx_smps*nx+sxy_smps*ny)*ux+(sxy_smps*nx+sy_smps*ny)*vx)

4 Select Boundaries 4, 6, and 13.

5 Specify additional expressions according to the following table:

NAME	EXPRESSION
W1	(sx_smps*ex_smps+sy_smps*ey_smps+2*sxy_smps*exy_smps)*Nx/2
Tdudx1	-((sx_smps*Nx+sxy_smps*Ny)*ux+(sxy_smps*Nx+sy_smps*Ny)*vx)

6 Select Boundaries 7, 9, and 11.

7 Specify additional expressions according to the following table:

NAME	EXPRESSION
W2	(sx_smps*ex_smps+sy_smps*ey_smps+2*sxy_smps*exy_smps)*Nx/2
Tdudx2	-((sx_smps*Nx+sxy_smps*Ny)*ux+(sxy_smps*Nx+sy_smps*Ny)*vx)

8 Click **OK**.

Scalar Expressions

Compute the J-integrals for each contour by summing the two integral expressions. In addition, define scalar expressions to compute the stress intensity factors.

1 Choose **Options>Expressions>Scalar Expressions**.

2 Specify expressions according to the following table. When finished, click **OK**.

NAME	EXPRESSION
`KI`	`sqrt(E_smps*abs(J))`
`J`	`2*(W+Tdudx)`
`KI1`	`sqrt(E_smps*abs(J1))`
`J1`	`2*(W1+Tdudx1)`
`KI2`	`sqrt(E_smps*abs(J2))`
`J2`	`2*(W2+Tdudx2)`

MESH GENERATION

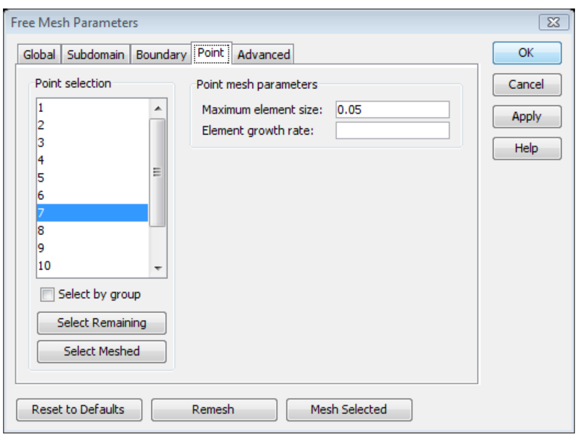
1 Choose **Mesh>Free Mesh Parameters**.

2 On the **Global** page, click the **Custom mesh size** option button.

3 In the **Maximum element size** edit field, type `0.4`.

4 Click the **Point** tab.

5 Select Point 7, then type `0.05` in the **Maximum element size** edit field.

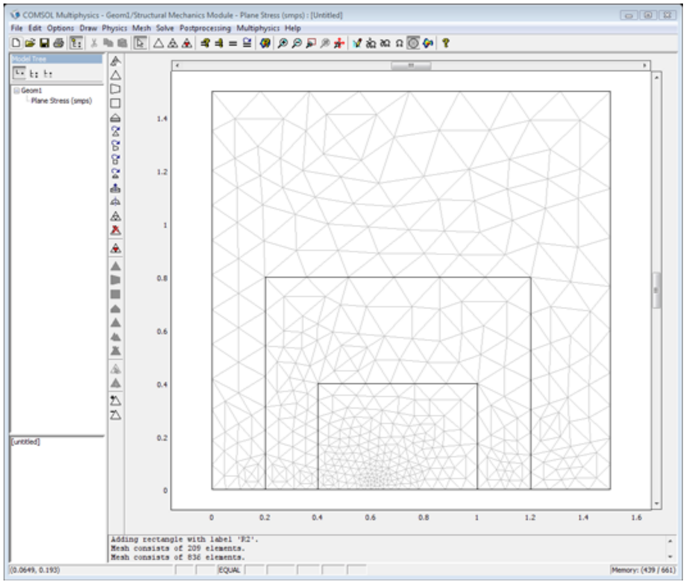


6 Click **OK**.

7 Click the **Refine Mesh** button on the Main toolbar.

This action first generates a mesh according to the specifications you gave in the **Free**

Mesh Parameters dialog box and then refines it once.



COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to start the analysis.

POSTPROCESSING AND VISUALIZATION

First display the stress intensity factor calculated with the outer contour:

1. From the **Postprocessing** menu, choose **Data Display>Global**.
2. In the **Expression** edit field type `KI`, then click **Apply**.

   The value for the stress intensity parameter KI appears in the message log.

   Next display the stress intensity factor values calculated by the two inner contours.
3. In the **Expression** edit field type `KI1`, then click **Apply**.
4. In the **Expression** edit field type `KI2`, then click **Apply**.

   The message log displays the stress intensity parameter values for the inner contours.

To plot the von Mises stress field and the deformed shape, follow these steps:

1. From the **Postprocessing** menu, choose **Plot Parameters**.
2. Click the **General** tab and select the **Surface**, **Deformed shape**, and **Geometry edges** check boxes only.

3 Click the **Surface** tab and select **von Mises stress** from the **Predefined quantities** list.

4 Click **OK**.

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

See Figure 11-2 for the von Mises stresses and the deformed shape plot.

12

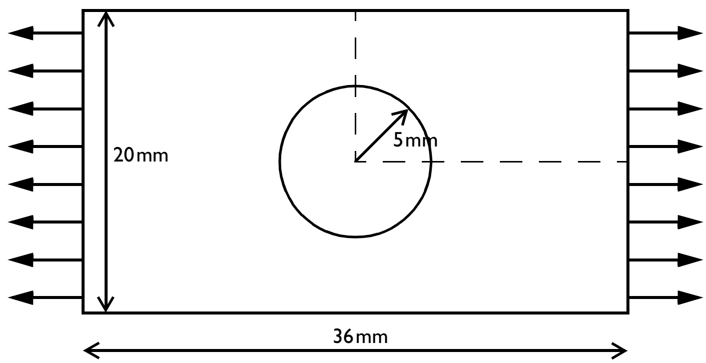
Nonlinear Material Models

This section contains examples of nonlinear material models that you can study using the application modes in the Structural Mechanics Module.

Elasto-Plastic Plate

Model Definition

In this example you analyze a plate with a hole when it is loaded above the elastic limit. The example is derived from section 7.10 of *The Finite Element Method* by O.C. Zienkiewicz (Ref. 1).



In addition to the original problem formulation, you follow the unloading history.

Because the plate is thin and the loads are in plane, a plane-stress condition can be assumed. Double symmetry means only a quarter of the plate needs to be analyzed.

MATERIAL

- Elastic properties: $E = 70000$ MPa and $\nu = 0.2$.
- Plastic properties: Yield stress 243 MPa and a linear isotropic hardening with tangent modulus 2171 MPa.

CONSTRAINTS AND LOADS

- Symmetry plane constraints are applied on the left most vertical boundary and the lower horizontal boundary.
- The right vertical edge is subjected to a stress, which increases from zero to a maximum value of 133.65 MPa and then is released again. The peak value is selected so that the mean stress over the section through the hole is 10% above the yield stress ($=1.1 \cdot 243 \cdot (20-10)/20$).

Results and Discussion

You can study the development of the plastic region in Figure 12-1. The parameter values are 0.59, 0.68, 0.78, 0.88, 0.98, and 1.08. These values are proportional to the load with parameter value 1.0 corresponding to the yield limit as average stress over the cross-section through the hole. For a material without strain hardening, the structure would thus have collapsed before reaching the final load level.

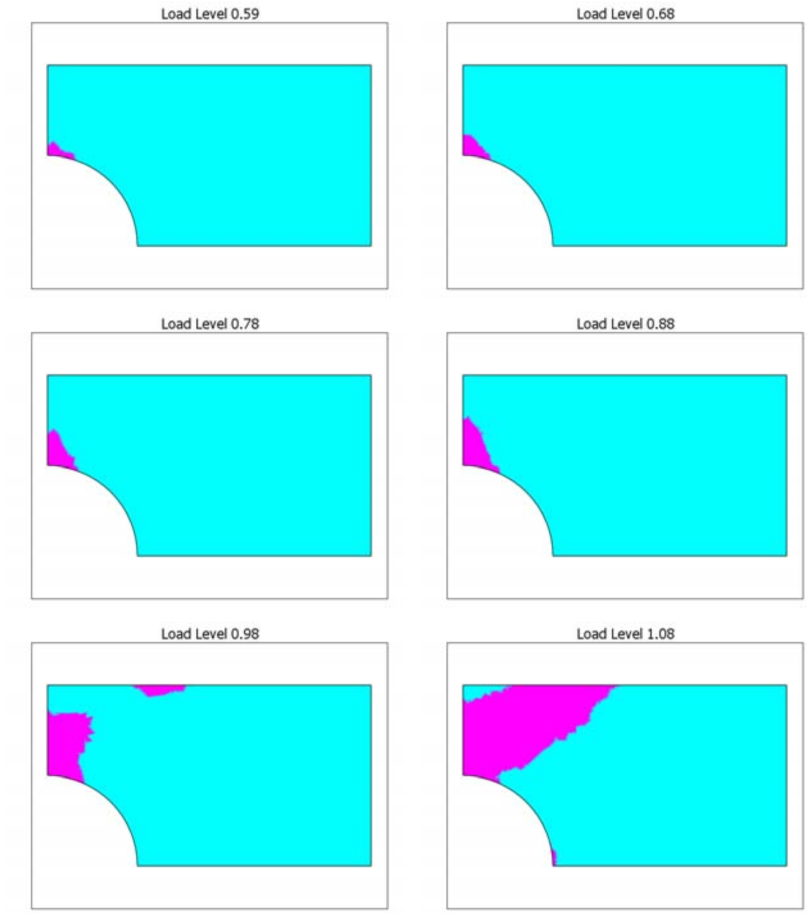


Figure 12-1: Development of plastic region (purple) with increased load.

Because an elasto-plastic solution is load-path dependent, it is important not to use too large steps in the load parameter when you anticipate a plastic flow. Usually you can take one large step up to the elastic limit, as this example shows. Moreover, reversed plastic flow can occur during the unloading. This is why small parameter steps are used at the end of the parameter range in this analysis.

Reference

1. O.C. Zienkiewicz and R.L. Taylor, *The Finite Element Method, Fourth Edition*, McGraw-Hill, 1991.

Model Library path: `Structural_Mechanics_Module/Nonlinear_Material_Models/elasto_plastic_plate`

Modeling Using the Graphical User Interface

1. Select **2D** in the **Space dimension** list on the **New** page in the **Model Navigator**.
2. Select **Structural Mechanics Module>Plane Stress>Static analysis elasto-plastic material**.
3. Click **OK**.

OPTIONS AND SETTINGS

1. Select **Model Settings** from the **Physics** menu to open the **Model Settings** dialog box.
2. Select **MPa** from the **Base unit system** list to use mm as length scale unit and MPa as stress unit. Click **OK**.

GEOMETRY MODELING

1. Shift-click the **Rectangle/Square** button on the Draw toolbar. In the **Width** edit field type `18` and in the **Height** edit field type `10`. Click **OK**.
2. Shift-click the **Ellipse/Circle (Centered)** button. In the **Radius** edit field type `5`, then click **OK**.
3. Select both objects by pressing Ctrl+A, then click the **Difference** button on the Draw toolbar to create the composite object CO1.
4. Click the **Zoom Extents** button on the Main toolbar.

PHYSICS SETTINGS

Boundary Settings

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

2 Specify boundary settings according to the following tables:

	BOUNDARIES 1, 3	
Page	Constraint	
	Constraint condition	Symmetry plane

	BOUNDARY 4	
Page	Load	
	F_x	`243/2*((para<1.1)*para+(para>=1.1)*(2.2-para))`
	Edge load is defined as force/area using the thickness	Selected

Subdomain Settings

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

2 Select Subdomain 1, then specify data on the **Material** page according to the following table:

SETTINGS	VALUE
E	`70000`
ν	`0.2`

3 Click the **Elasto-plastic material data** button to open the **Elasto-Plastic Material Settings** dialog box.

4 In the σ_{ys} edit field type `243` and in the E_{Tiso} edit field type `2171`.

5 Click **OK**.

The Gauss point order defaults to 2 times the order of the shape function. When solving a static problem with second-order triangular elements, you can improve the performance by lowering the Gauss point order from 4 to 2. For a linear elastic problem this does not make a large difference, but when solving an elasto-plastic problem the difference in memory and speed are much larger, because the solver computes the plastic strain in the Gauss points of the elements.

6 Type `2 2 2` in the **gporder** edit field on the **Element** page.

Note: Lowering the Gauss point order to the same value as the shape function order does not work together with quad and hex elements.

7 Click **OK to** close the **Subdomain Settings** dialog box.

MESH GENERATION

1 Click the **Initialize Mesh** button on the Main toolbar to generate the mesh.

2 Click the **Refine Mesh** button to refine the mesh.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 Type the name `para` in the **Parameter name** edit field.

3 Enter `0 0.44:0.05:0.59 0.63:0.05:1.08 1.1:0.2:1.9 1.95:0.05:2.2` in the **Parameter values** edit field. Click **OK**.

4 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

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

2 On the **General** page select **0.59** from the **Parameter value** list.

3 Click the **Surface** tab.

4 Type `epe_smps>0` in the **Expression** edit field to get a flag for plastic deformation.

5 Select **cool** from the **Colormap** list.

6 Return to the **General** page and step through the values in the **Parameter value** list, clicking **Apply** in between to generate the plots.

Viscoelastic Material

Analysis of Viscoelastic Materials

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.

Linear viscoelastic material models are expressed in integral-equation (Equation 12-1) or differential-equation (Equation 12-2) models.

$$\sigma = \int_0^t G(t-t')\frac{\partial\varepsilon}{\partial t'}dt' \qquad (12\text{-}1)$$

$$\sum_i a_i \frac{\partial^i \sigma}{\partial t^i} = \sum_k b_k \frac{\partial^k \varepsilon}{\partial t^k} \qquad (12\text{-}2)$$

Models of the general type indicated in Equation 12-2 tend to require high-order derivatives and are less suitable for a finite element implementation.

The function $G(t)$ in Equation 12-1 is the *relaxation modulus function* and can be measured as the stress when the material is held at a constant strain. This function is often approximated in a Prony series as shown in Equation 12-3.

$$G(t) = G_0\left[\mu_0 + \sum_{m=1}^{M} \mu_m e^{\frac{-t}{\lambda_m}}\right] \qquad (12\text{-}3)$$

$$\sum_{m=0}^{M} \mu_m = 1 \qquad (12\text{-}4)$$

Here G_0, μ_m, and λ_m are material constants.

It is usually assumed that the viscous part of the deformation is incompressible, so that the volume change is purely elastic. With this assumption and the relaxation modulus represented by Equation 12-3, a differential equation form which is well suited for

COMSOL Multiphysics can be developed. It is given in Equation 12-5 through Equation 12-10.

$$s = 2G_0\left(\mu_0 e + \sum_{m=1}^{M} \mu_m q_m\right) \tag{12-5}$$

$$\sum_{m=0}^{M} \mu_m = 1 \tag{12-6}$$

$$\dot{q}_m + \frac{1}{\lambda_m} q_m = \dot{e} \tag{12-7}$$

$$\sigma = s - pI \tag{12-8}$$

$$p = -K \cdot \mathtt{trace}(\varepsilon) \tag{12-9}$$

$$e = \varepsilon - \frac{1}{3}\mathtt{trace}(\varepsilon) \cdot I \tag{12-10}$$

The physical interpretation of this model is that the components of the stress and strain deviators, s and e, are related through a *generalized Maxwell model*, as shown in Figure 12-2. The instantaneous shear modulus is G_0, the long term shear modulus is $\mu_0 G_0$, and the bulk modulus is K. Note that each of the variables q_m has as many components as the number of strain components of the problem class. The coefficient μ_m can be interpreted as the relative stiffness of the spring in branch m of the generalized Maxwell model. Each time constant λ_m is the time constant of the spring-dashpot pair in the same branch. Following the analogy, the abstract variable q_m is the extension of the corresponding spring.

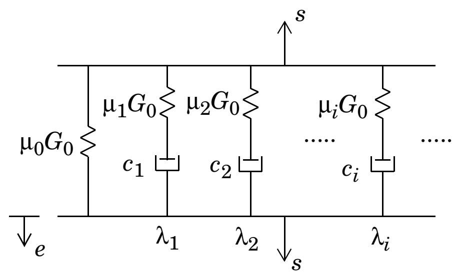


Figure 12-2: Generalized Maxwell model.

When the viscous strains grow large, the near incompressibility can cause numerical problems if only displacements are used in the interpolating functions. In such cases, use a so-called mixed formulation. In a mixed formulation, the pressure, p, is introduced as an independent variable, while equation Equation 12-9 is only fulfilled in a weak sense.

The interpolation functions used for the variables q_m and the pressure (if used) should be one order lower than what is used for the displacements, because these variables are added to the strains and stresses computed from displacement derivatives.

For many materials, the viscoelastic properties have a strong dependence on the temperature. A common assumption is that the material is *thermorheologically simple* (TRS). In a material of this class, a change in the temperature can be transformed directly into a change in the time scale. The reduced time τ is defined as

$$\tau = \int_0^t \frac{dt'}{a_T(T(t'))} \tag{12-11}$$

where $a_T(T)$ is a shift function. The implication is that the problem can be solved using the original material data, provided that the time is transformed into the reduced time. One commonly used shift function is defined by the WLF (Williams-Landel-Ferry) equation:

$$\log(a_T) = \frac{-C_1(T-T_0)}{C_2+(T-T_0)} \tag{12-12}$$

Note that $a_T(T_0)$=1, so that T_0 is the temperature at which the original material data is given. Usually T_0 is taken as the *glass temperature* of the material. If the temperature drops below T_0–C_2, the WLF equation is no longer valid. The constants C_1 and C_2 are material dependent, but with T_0 as the glass temperature the values $C_1 = 17.4$ and $C_2 = 51.6$ K are reasonable approximations for many polymers if a base-10 logarithm is assumed.

If you think of the shift function as a multiplier to the viscosity in each of the dashpots in the generalized Maxwell model, Equation 12-7 will for a TRS material be modified to

$$\dot{q}_m + \frac{1}{a_T(T(t))\lambda_m} q_m = \dot{e} \qquad (12\text{-}13)$$

Model Definition

The geometry is a long thick-walled cylinder with an inner radius of 5 mm and an outer radius of 10 mm. The inner surface is subjected to a prescribed radial displacement of 0.001 mm. The model shows the decay of the stresses during a period of two hours.

A quarter of the cylinder is modeled. The first versions of the problem are axially symmetric, while the subsequent versions are not.

The example is developed in four steps:

1. A pure Maxwell model (only one spring in series with a dashpot) is used.
2. The material model is changed to a four-term generalized Maxwell model.
3. A stationary temperature field is added, causing the problem to lose its axisymmetry.
4. The temperature field is also transient.

When you solve this type of problem, where a load is applied instantaneously at the beginning of the analysis, you can choose between two approaches: Either apply the load over a short period of time at the beginning of the time stepping, or use a separate static analysis to obtain the initial conditions. Here the latter method is used. It will be necessary to create correct initial conditions for the q_m variables, that is, they must be set equal to the corresponding values of the strain deviator components obtained in the initial step.

Note: The q variables use linear discontinuous shape functions (`shdisc`), because they are only a part of the material model and do not represent a field that must be continuos. This increases the number of degrees of freedom, but there is no coupling between elements, resulting in low bandwidth of the matrices. The elements are linear, because they appear in equations with strains that come from the derivative of a second-order displacement field.

MATERIAL

- Elastic (instantaneous) data: Isotropic with E = 67 GPa = $67 \cdot 10^3$ MPa, ν = 0.22.
- Viscoelastic data, first analysis: Shear modulus behaves as a Maxwell material with time constant 1 h, giving μ_1 = 1 and λ_1 = 3600 s.
- Viscoelastic data, following analyses: Four-term Generalized Maxwell material with:
  - μ_1 = 0.04, λ_1 = 20 s
  - μ_2 = 0.06, λ_2 = 300 s
  - μ_3 = 0.09, λ_3 = 3000 s
  - μ_4 = 0.25, λ_4 = 12000 s
- Thermal properties: A WLF model is used with C_1=17.44 and C_2=51.6. The reference temperature is 500 K.
- Heat conduction properties: ρ = 1100 kg/m^3 = $1.1 \cdot 10^{-9}$ t/mm^3, c_p = 2100 J/kg, K = $2100 \cdot 10^6$ Nmm/(t·K), k = $6 \cdot 10^{-2}$ W/(m·K) = $6 \cdot 10^{-2}$ Nmm/(s·mm·K).

CONSTRAINTS

- The circumferential displacements are constrained on the radial edges.
- The inner edge of the hole is constrained to a radial displacement of 0.001 mm.

HEAT TRANSFER BOUNDARY CONDITIONS

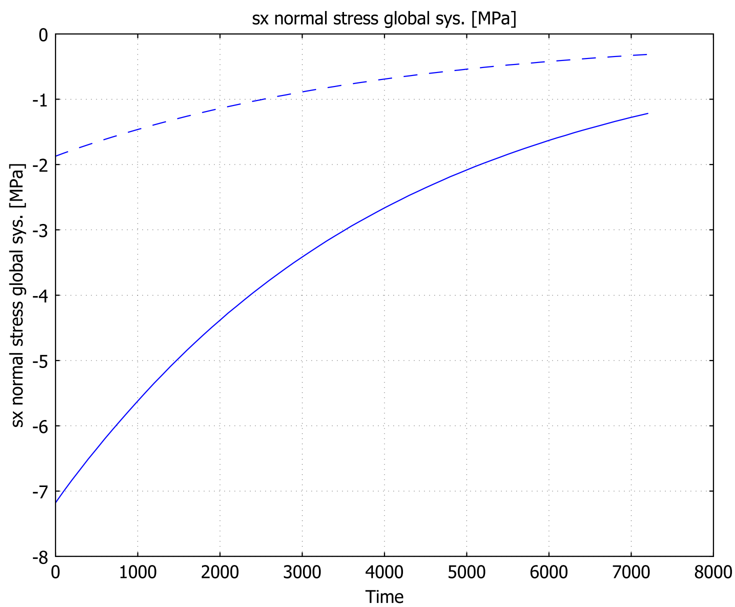
- Stationary analysis: The inner and outer circular edge has a temperature distribution varying linearly with the y-coordinate from 500 K at the y = 0 symmetry section to 505 K at the x = 0 symmetry section.
- Transient analysis: The temperature distribution obtained from the stationary analysis is used as initial condition, while all edges are insulated.

Results and Discussion

The first version of the problem can be solved analytically through the correspondence principle. Using such a solution, the radial stress at the inner edge has been evaluated. It is compared with the COMSOL Multiphysics solution in the table below.

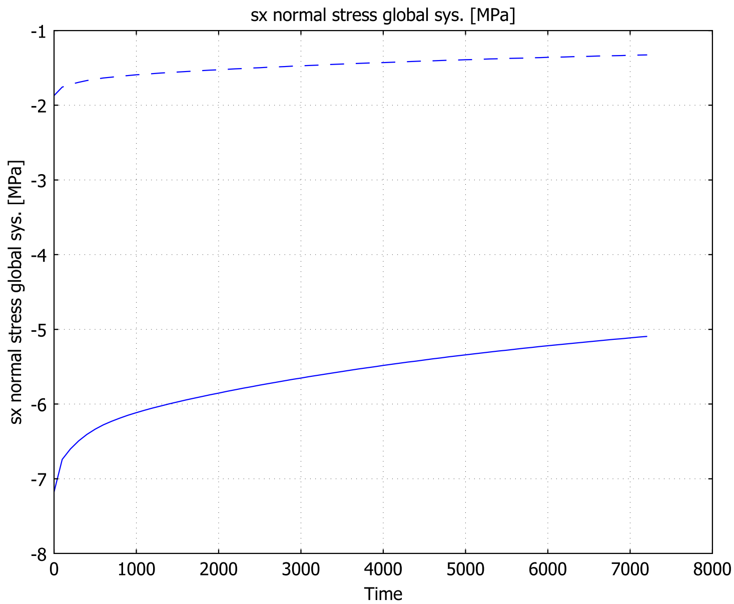
TIME	COMSOL MULTIPHYSICS	ANALYTICAL
0	-7.22 MPa	-7.23 MPa
3600 s	-2.96 MPa	-2.97 MPa
7200 s	-1.24 MPa	-1.24 MPa

The evolution of the radial stress at r = 5 mm (solid) and r = 7.5 mm (dashed) is shown in the figure below.

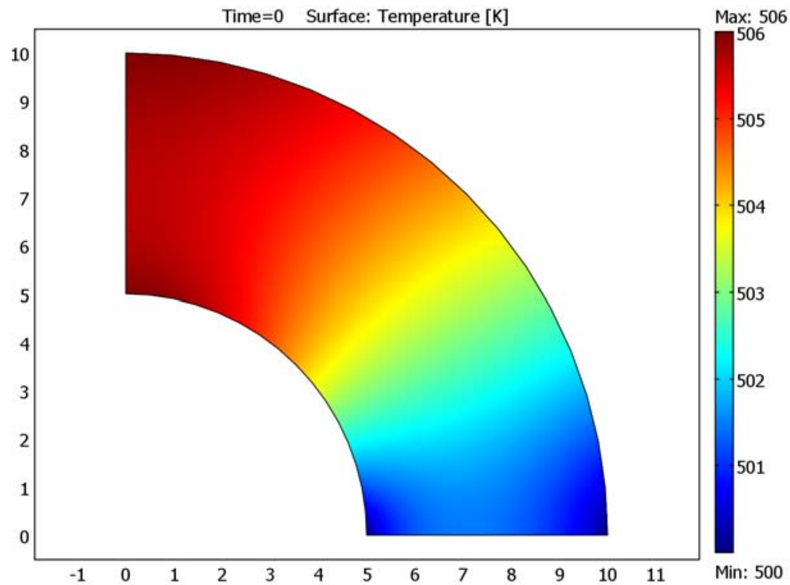


In this case no problems occurred with incompressibility constraints. You can see from the analytical solution that the effective Poisson's ratio increased from the initial value of 0.22 to about 0.45.

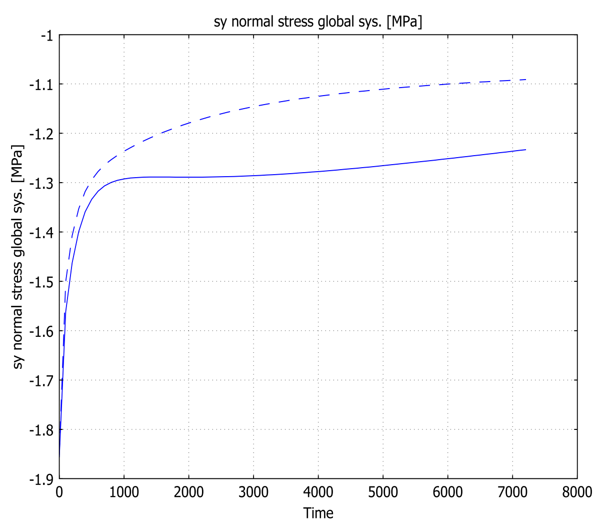
When you use a less idealized material model (the one with four time constants), the stress decay looks quite different as is shown below. One major difference is that the coefficient μ_0 is no longer zero, because it is in a the pure Maxwell model. As a consequence there are substantial stresses even after a long time.



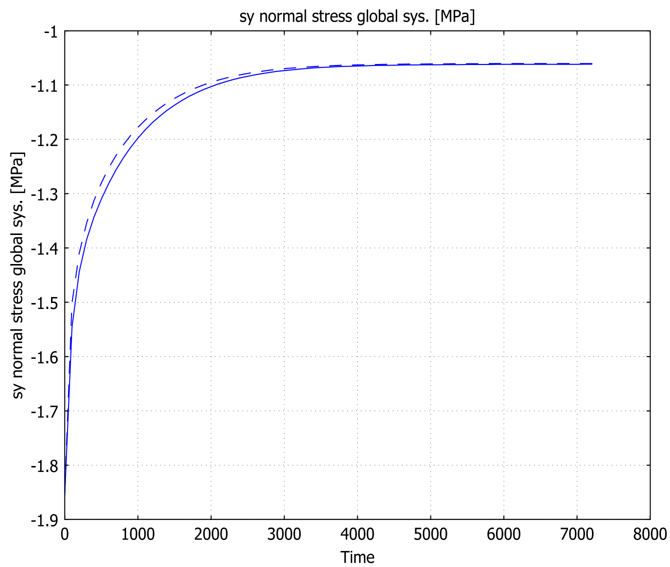
When the temperature field shown below is introduced, the relaxation will be faster where the temperature is higher.



In the plot below, the radial stresses at radius 7.5 mm display this effect. The solid line is taken at $y = 0$ (cold) and the dashed line at $x = 0$ (warm).



In the last analysis, the temperature initially has the skew distribution above, but is allowed to redistribute to a final homogeneous value. Again, we compare the radial stresses at radius 7.5 mm. The initial behavior is similar to the previous case, but as the difference in material properties decreases, the curves approach each other. The strain rate in the initially warm point decreases, while it increases in the initially cold point.



Model Library path: `Structural_Mechanics_Module/Nonlinear_Material_Models/viscoelastic_material`

Modeling Using the Graphical User Interface

STEP 1—A PURE MAXWELL MODEL

Model Navigator

1 Select **2D** in the **Space dimension** list on the **New** page in the **Model Navigator**.

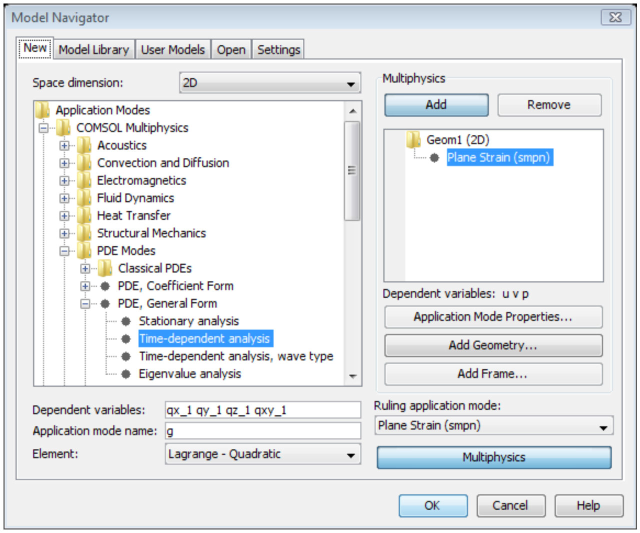
2 Select **Structural Mechanics Module>Plane Strain>Quasi-static analysis**.

3 Click the **Multiphysics** button, then click the **Add** button.

4 Select **COMSOL Multiphysics**>**PDE Modes**>**PDE, General Form**>**Time-dependent analysis.**

5 In the **Dependent variables** edit field type `qx_1 qy_1 qz_1 qxy_1`.

6 In the **Application mode name** edit field type `visc_1`.



7 Click the **Add** button, then click **OK** to close the **Model Navigator**.

Options and Settings

1 Select **Model Settings** from the **Physics** menu to open the **Model Settings** dialog box.

2 Select **MPa** from the **Base unit system** list to use mm as length scale unit and MPa as stress unit. Click **OK**.

3 Select **Axes/Grid Settings** from the **Options** menu and give axis and grid settings according to the following table:

AXIS		GRID	
x min	-12	x spacing	1
x max	12	Extra x	
y min	-12	y spacing	1
y max	12	Extra y	

4 When finished, click **OK**.

5 Select **Constants** from the **Options** menu and enter constant names, expressions, and descriptions (optional) according to the following table:

NAME	EXPRESSION	DESCRIPTION	UNIT
`mu_1`	`1`	Prony series coefficient	
`mu_0`	`1-mu_1`	Prony series coefficient	
`lambda_1`	`3600`	Prony series time constant	s

Note that the unit does not appear in the dialog box.

6 Click **OK**.

7 Select **Expressions>Scalar Expressions** from the **Options** menu and enter expression names, expressions, and descriptions (optional) according to the following table:

NAME	EXPRESSION	DESCRIPTION	UNIT
`kappa`	`E_smpn/(3*(1-2*nu_smpn))`	Instantaneous bulk modulus	MPa
`G`	`E_smpn/(2*(1+nu_smpn))`	Instantaneous shear modulus	MPa
`ekk`	`ex_smpn+ey_smpn`	Trace of strain tensor	
`p2`	`-kappa*ekk`	Pressure	MPa
`mu_q_x`	`mu_1*qx_1`	Sum of terms in Prony series	
`mu_q_y`	`mu_1*qy_1`	Sum of terms in Prony series	
`mu_q_z`	`mu_1*qz_1`	Sum of terms in Prony series	
`mu_q_xy`	`mu_1*qxy_1`	Sum of terms in Prony series	

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

Geometry Modeling

1 Click the **Ellipse/Circle (Centered)** button on the Draw toolbar. Click the left mouse button at (0, 0), then move the mouse to (10, 10) and click the left mouse button again to create a circle with radius equal to the cylinder's outer radius.

2 Click the **Ellipse/Circle (Centered)** toolbar button. Click the left mouse button at (0, 0), then move the mouse to (5, 5) and click the left mouse button again to create a circle with radius equal to the cylinder's inner radius.

3 Select both circles, then click the **Difference** button on the Draw toolbar.

4 Click the **Rectangle/Square** button on the Draw toolbar. Click the left mouse button at (0, 0), then move the mouse to (11, 11) and click the left mouse button again.

5 Select both solid objects and click the **Intersection** button on the Draw toolbar.

Physics Settings

Boundary Settings

Constrain the displacements of the cuts through the cylinder to have symmetry conditions. Specify the radial displacement on the inner surface to be 0.001 mm. The viscoelastic variables should not be prescribed on any boundaries as they are by default.

1 Select **Plane Strain** from the **Multiphysics** menu.
2 Select **Boundary Settings** from the **Physics** menu.
3 On the **Constraint** page, specify settings according to the following table:

SETTINGS	BOUNDARIES 1, 2	BOUNDARY 3
Constraint condition	Symmetry plane	Prescribed displacement
Coordinate system	-	Tangent and normal coord. sys. (t,n)
R_n	-	`-0.001`

4 Click **OK**.
5 Select **PDE, General Form** from the **Multiphysics** menu.
6 Select **Boundary Settings** from the **Physics** menu.
7 On the **Type** page, specify boundary settings according to the following table:

SETTING	BOUNDARIES 1–4
Boundary condition type	Neumann boundary condition

8 Click **OK**.

Subdomain Settings

1 Select **Plane Strain** from the **Multiphysics** menu.
2 Select **Subdomain Settings** from the **Physics** menu.
3 On the **Material** page, specify the material properties of the cylinder according to the following table:

SETTINGS	SUBDOMAIN 1
Material model	Isotropic material
E	`67e3`
ν	`0.22`

4 Click **OK**.

5 Select **PDE, General Form** from the **Multiphysics** menu.

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

7 Change the shape functions to be discontinuous of order 1, because they do not require continuity. Constrain the q variables to the initial strain deviator. The changes are summarized in the following table:

PAGE	SETTING	SUBDOMAIN 1
Γ		
	Flux vector	`0 0` `0 0` `0 0` `0 0`
F		

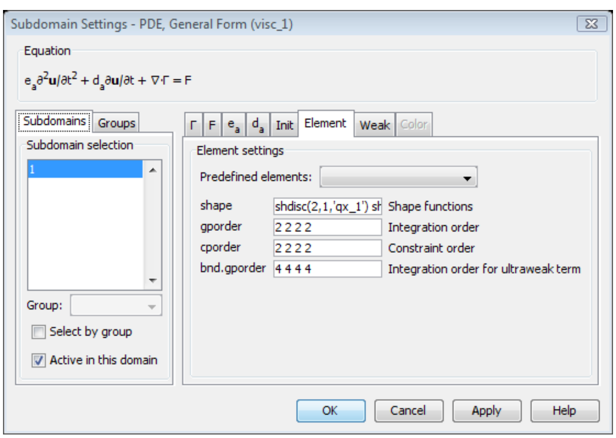
	Source term	`qx_1-(ex_smpn-ekk/3)`
		`qy_1-(ey_smpn-ekk/3)`
		`qz_1-(-ekk/3)`
		`qxy_1-exy_smpn`

Element		

	shape	`shdisc(2,1,'qx_1')`
		`shdisc(2,1,'qy_1')`
		`shdisc(2,1,'qz_1')`
		`shdisc(2,1,'qxy_1')`

	gporder	`2 2 2 2`

8 When finished, click **OK**.



9 Select **Equation System>Subdomain Settings** from the **Physics** menu.

10 On the **Variables** page, change the definitions of the stresses according to the following table:

SETTINGS	SUBDOMAIN 1
sx_smpn	2*G*(mu_0*(ex_smpn-ekk/3)+mu_q_x)-p2
sy_smpn	2*G*(mu_0*(ey_smpn-ekk/3)+mu_q_y)-p2
sz_smpn	2*G*(mu_0*(-ekk/3)+mu_q_z)-p2
sxy_smpn	2*G*(mu_0*exy_smpn+mu_q_xy)

11 When finished, click **OK** to close the dialog box.

Mesh Generation

1. Select **Free Mesh Parameters** from the **Mesh** menu.
2. Select **Normal** in the **Predefined mesh sizes** list.
3. Click the **Boundary** tab. Set **Maximum element size** to `0.4` for Edge 3.
4. Click the **Remesh** button, then click **OK**.
5. Click the **Zoom Extents** button on the Main toolbar to examine the mesh.

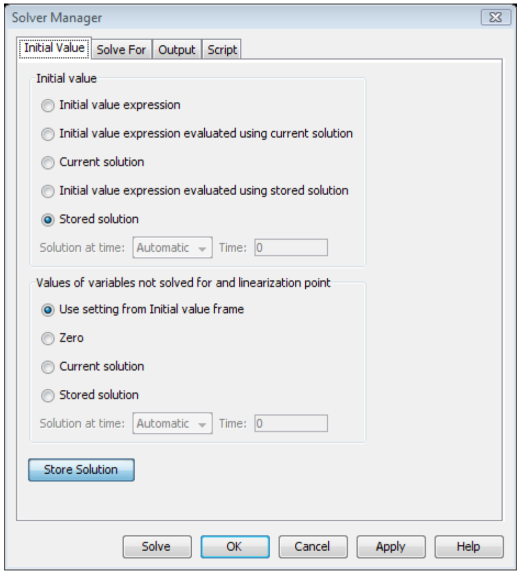
Computing the Initial Conditions

1. Select **Solver Parameters** from the **Solve** menu.
2. Select **Stationary** in the **Solver** list, then click **OK** to close the dialog box.
3. Click the **Solve** button on the Main toolbar.

Initial Conditions for the Transient Solution

1. Select **Solver Manager** from the **Solve** menu.
2. Select **Stored solution** as initial value.

3 Click the **Store Solution** button to save the current solution to be used as initial condition.



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

Physics Settings for the Time-Dependent Problem

1 Select **PDE, General Form** from the **Multiphysics** menu.

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

3 On the **F** page, add the true time-dependent formulation according to the following table:

SETTINGS	SUBDOMAIN 1
Source term	`-qx_1/lambda_1-qy_1/lambda_1 -qz_1/lambda_1-qxy_1/lambda_1`
Weak	
dweak	`-qx_1_test*(ex_smpn_time-ekk_time/3) -qy_1_test*(ey_smpn_time-ekk_time/3) -qz_1_test*(-ekk_time/3) -qxy_1_test*(exy_smpn_time)`

4 When finished, click **OK** to close the dialog box.

Computing the Solution for the Time-Dependent Problem

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

2 Select **Time dependent** in the **Solver** list.

3 On the **General** page, specify the solver parameters according to the following table:

EDIT FIELD	VALUE
Times	`0:100:7200`
Absolute tolerance	`1e-8`

4 Click **OK** to close the dialog box, then click the **Solve** button on the Main toolbar.

Postprocessing and Visualization

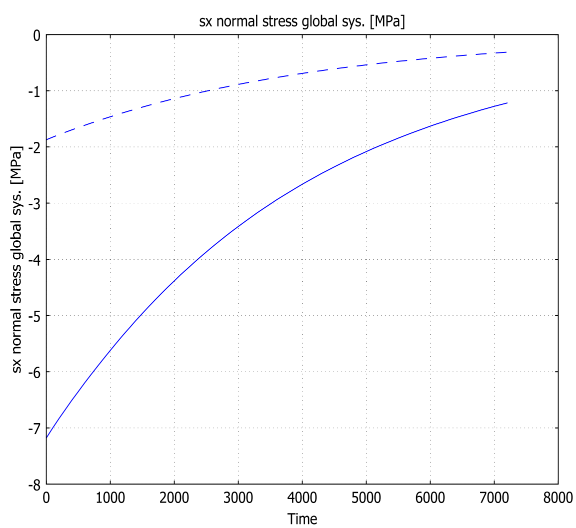
Produce a graph of the radial stress as function of time at $r = 5$ mm and $r = 7.5$ mm.

1 Select **Cross-Section Plot Parameters** from the **Postprocessing** menu.

2 On the **Point** page, enter the following parameters:

EDIT FIELD	VALUE
Predefined quantities	Plane Strain (smpn)>sx normal stress global sys.
x	`5 7.5`
y	`0 0`

3 Click **OK** to display the graphs.



STEP 2—ADDING MORE TERMS IN THE PRONY SERIES

The most structured method to enter the input data is to add one new application mode for each term in the Prony series. The only connection between the variables are supplied through the scalar expressions.

Model Navigator

1 Choose **Model Navigator** from the **Multiphysics** menu.

2 Select **2D** in the **Space dimension** list on the **New** page in the **Model Navigator**.

3 Select **COMSOL Multiphysics>PDE Modes>PDE, General Form>Time-dependent analysis.**

4 In the **Dependent variables** edit field type `qx_2 qy_2 qz_2 qxy_2`.

5 In the **Application mode name** edit field type `visc_2`.

6 Click the **Add** button.

7 In the **Dependent variables** edit field type `qx_3 qy_3 qz_3 qxy_3`.

8 In the **Application mode name** edit field type `visc_3`.

9 Click **Add**.

10 In the **Dependent variables** edit field type `qx_4 qy_4 qz_4 qxy_4`.

11 In the **Application mode name** field type `visc_4`.

12 Click the **Add** button, then close the **Model Navigator** by clicking **OK**.

Options and Settings

1 Select **Constants** from the **Options** menu and edit constant names, expressions, and descriptions (optional) according to the following table:

NAME	EXPRESSION	DESCRIPTION	UNIT
mu_1	0.04	Prony series coefficient 1	
mu_2	0.08	Prony series coefficient 2	
mu_3	0.09	Prony series coefficient 3	
mu_4	0.25	Prony series coefficient 4	
mu_0	1-mu_1-mu_2-mu_3-mu_4	Prony series coefficient 0	
lambda_1	20	Prony series time constant 1	s
lambda_2	300	Prony series time constant 2	s
lambda_3	3000	Prony series time constant 3	s
lambda_4	12000	Prony series time constant 4	s

Note that the unit does not appear in the dialog box.

2 Select **Expressions>Scalar Expressions** from the **Options** menu and modify expressions according to the following table:

NAME	EXPRESSION	DESCRIPTION
`mu_q_x`	`mu_1*qx_1+mu_2*qx_2+mu_3*qx_3+mu_4*qx_4`	Sum of terms in Prony series
`mu_q_y`	`mu_1*qy_1+mu_2*qy_2+mu_3*qy_3+mu_4*qy_4`	Sum of terms in Prony series
`mu_q_z`	`mu_1*qz_1+mu_2*qz_2+mu_3*qz_3+mu_4*qz_4`	Sum of terms in Prony series
`mu_q_xy`	`mu_1*qxy_1+mu_2*qxy_2+mu_3*qxy_3+mu_4*qxy_4`	Sum of terms in Prony series

Physics Settings

Boundary Settings

The boundary conditions must be removed for the new variables.

1 Select **PDE, General Form (visc_2)** from the **Multiphysics** menu

2 Select **Boundary Settings** from the **Physics** menu.

3 On the **Type** page, specify boundary settings according to the following table:

SETTING	BOUNDARIES 1–4
Boundary condition type	Neumann boundary condition

4 Do the same steps for `visc_3` and `visc_4`.

Subdomain Settings

1 Select **PDE, General Form (visc_2)** from the **Multiphysics** menu.

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

3 Set the values according to the following tables:

PAGE	SETTINGS	SUBDOMAIN 1
Γ		
	Flux vector	0 0 0 0 0 0 0 0
F		

PAGE	SETTINGS	SUBDOMAIN 1

	Source term	`qx_2-(ex_smpn-ekk/3)`
		`qy_2-(ey_smpn-ekk/3)`
		`qz_2-(-ekk/3)`
		`qxy_2-exy_smpn`

Element		

	shape	`shdisc(2,1,'qx_2')`
		`shdisc(2,1,'qy_2')`
		`shdisc(2,1,'qz_2')`
		`shdisc(2,1,'qxy_2')`

	gporder	`2 2 2 2`
Weak		

	dweak	`-qx_2_test*(ex_smpn_time-ekk_time/3)`
		`-qy_2_test*(ey_smpn_time-ekk_time/3)`
		`-qz_2_test*(-ekk_time/3)`
		`-qxy_2_test*(exy_smpn_time)`

4 Do the same steps for `visc_3` and `visc_4`, substituting all `_2` with `_3` and `_4`, respectively.

5 For `visc_1`, reset the **Source** `term` on the **F** page because you must generate new initial conditions. The other data is already correct.

SETTING	SUBDOMAIN 1

Source term	`qx_1-(ex_smpn-ekk/3)`
	`qy_1-(ey_smpn-ekk/3)`
	`qz_1-(-ekk/3)`
	`qxy_1-exy_smpn`

Computing the Initial Conditions

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

2 Select **Stationary** in the **Solver** list, then click **OK**.

3 Click the **Solve** toolbar button.

Initial Conditions for the Time-Dependent Problem

1 From the **Solve** menu select **Solver Manager**.

2 Click the **Store Solution** button to save the current solution to be used as initial condition. Click **OK**.

Physics Settings for the Time-Dependent Problem

1 Select **PDE, General Form (visc_1)** from the **Multiphysics** menu

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

3 On the **F** page, add the true time-dependent formulation according to the following table:

SETTING	SUBDOMAIN 1
Source term	`-qx_1/lambda_1-qy_1/lambda_1-qz_1/lambda_1-qxy_1/lambda_1`

4 Do the same steps for `visc_2`, `visc_3`, and `visc_4`, substituting all instances of `_1` with `_2`, `_3`, and `_4`, respectively.

Computing the Solution for the Time-Dependent Problem

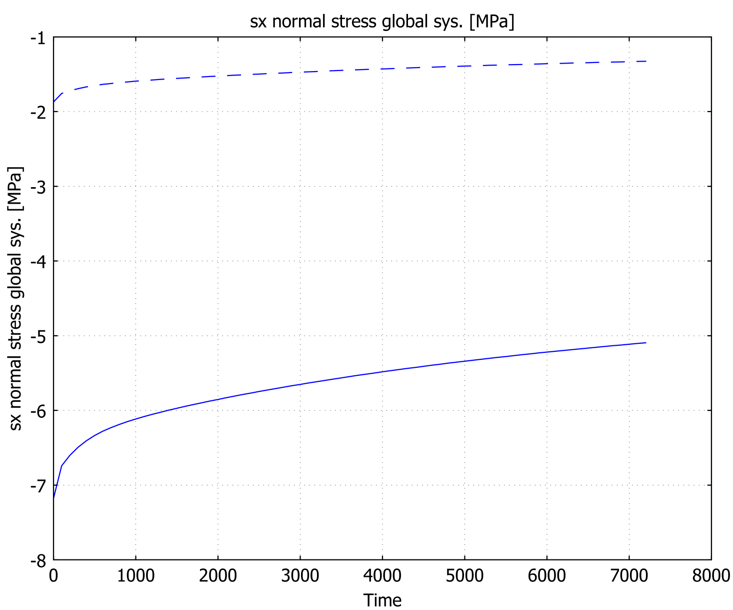
1 Select **Solver Parameters** from the **Solve** menu.

2 Select **Time dependent** in the **Solver** list, then click **OK**.

3 Click the **Solve** button on the Main toolbar.

Postprocessing and Visualization

Produce a graph of the radial stress as function of time at $r = 5$ and $r = 7.5$.



STEP 3—ADD A TEMPERATURE DEPENDENCE

To solve the thermal problem, you must add a heat equation and modify the time constants in the Prony series.

Model Navigator

1. Select **Model Navigator** from the **Multiphysics** menu.
2. Select **2D** from the **Space dimension** list on the **New** page in the **Model Navigator**.
3. Select **COMSOL Multiphysics>Heat Transfer>Conduction>Transient analysis**.
4. Click the **Add** button, then close the **Model Navigator** by clicking **OK**.

Options and Settings

1. Select **Constants** from the **Options** menu and add constant names, expressions, and descriptions (optional) according to the following table:

NAME	EXPRESSION	DESCRIPTION	UNIT
C1	17.44	WLF constant C1	
C2	51.6	WLF constant C2	K
T_ref	500	WLF reference temperature	K

Note that the unit does not appear in the dialog box.

2. Select **Expressions>Scalar Expressions** from the **Options** menu and add the expression for the WLF shift factor:

NAME	EXPRESSION	DESCRIPTION	UNIT
time_shift	10^(-C1*(T-T_ref)/(C2+(T-T_ref)))	WLF time shift factor	1

Physics Settings

Boundary Settings

1. Select **Heat Transfer by Conduction** from the **Multiphysics** menu.
2. Select **Boundary Settings** from the **Physics** menu.
3. Specify boundary settings according to the following table:

SETTINGS	BOUNDARY 3	BOUNDARY 4
Boundary condition	Temperature	Temperature
T_0	500+y/5*6	500+y/10*6

Subdomain Settings

You must specify the material data for the thermal problem. In addition, the equations for the viscoelastic variables must be reset for initial value computation.

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

2 On the **Physics** page, specify the thermal properties according to the following table:

SETTINGS	SUBDOMAIN 1
Physics	
k (isotropic)	`0.06`
ρ	`1.1e-9`
c_p	`2.1e9`

3 Select **PDE, General Form (visc_1)** from the **Multiphysics** menu.

4 Reset all **F** terms for the viscoelastic variables for initial value generation in the same way as in previous steps for `visc_1`,

SETTING	SUBDOMAIN 1

Source term	`qx_1-(ex_smpn-ekk/3)`
	`qy_1-(ey_smpn-ekk/3)`
	`qz_1-(-ekk/3)`
	`qxy_1-exy_smpn`

5 Repeat the procedure for `visc_2`, `visc_3`, and `visc_4` with the previously described substitutions of `_1`.

Computing the Initial Conditions

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

2 Select **Stationary** in the **Solver** list, then click **OK**.

3 Click the **Solve** button on the Main toolbar.

Initial Conditions for the Time-Dependent problem

1 Select **Solver Manager** from the **Solve** menu.

2 Click the **Store Solution** button to save the current solution to be used as initial condition, then click **OK**.

Physics Settings for the Time-Dependent Problem

1 Select **PDE, General Form (visc_1)** from the **Multiphysics** menu.

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

3 On the **F** page, add the true time-dependent formulation, including temperature-dependent time shift, according to the following table:

SETTING	SUBDOMAIN 1

Source term	`-qx_1/lambda_1/time_shift`
	`-qy_1/lambda_1/time_shift`
	`-qz_1/lambda_1/time_shift`
	`-qxy_1/lambda_1/time_shift`

4 Do the same steps for `visc_2`, `visc_3`, and `visc_4`, substituting all `_1` with `_2`, `_3`, and `_4`, respectively.

Computing the Solution for the Time-Dependent Problem

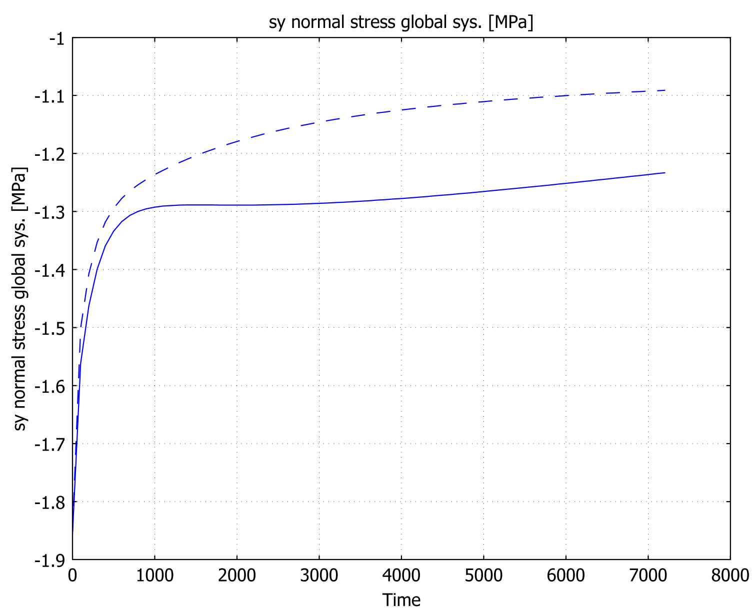
1 Select **Solver Parameters** from the **Solve** menu.

2 Select **Time dependent** in the **Solver** list, then click **OK**.

3 Click the **Solve** button on the Main toolbar.

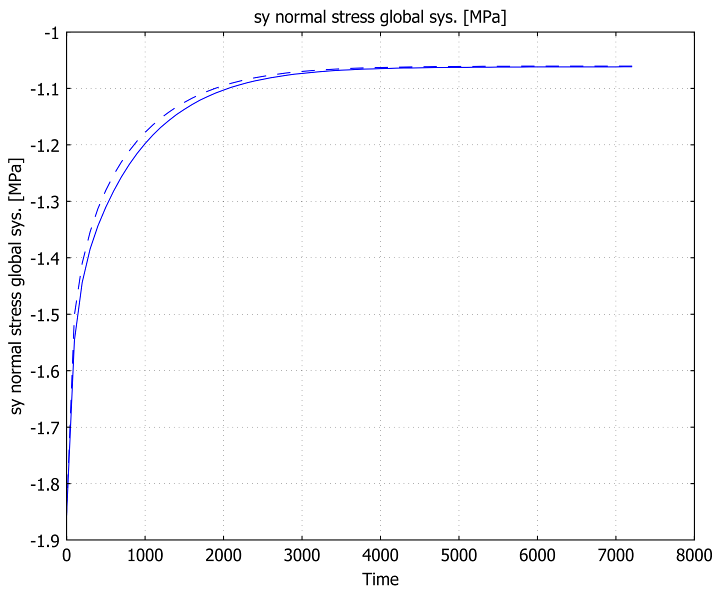
Postprocessing and Visualization

Produce a graph of the radial stress as function of time at (7.5, 0) and (0, 7.5).



STEP 4—SOLUTION WITH A TRANSIENT TEMPERATURE

In the last version of the problem the temperature field is allowed to settle to a steady state using insulated conditions. The initial temperature is the same as above. The only change needed is thus to set the boundary conditions for the heat transfer problem to be thermally insulated also for Boundaries 3 and 4. Solve again and produce a plot of the same type as above.



Hyperelastic Seal

Introduction

In this model you study the force-deflection relation of a car door seal made from a soft rubber material. The model uses a hyperelastic material model together with formulations that can account for the large deformations.

THEORETICAL BACKGROUND

You find the theory for hyperelastic materials in the section "Hyperelastic Materials" on page 176 in the *Structural Mechanics Module User's Guide*.

Model Definition

The seal is compressed between a stationary plane surface and an indenting cylinder. It is of special interest to investigate the effect of air confined within the seal. Figure 12-3 below shows the geometry of the seal.

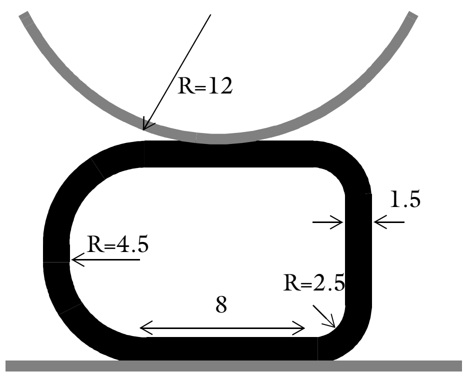


Figure 12-3: Model geometry.

The model describes a cross section of the seal with an assumption of plane strain. The contacting surfaces are rigid when compared to the seal.

When computing the pressure from the air compressed inside the seal, the current cross-section area is required. One useful method for computing an area is by using Gauss' theorem, and converting to a contour integral.

$$A = \int 1 da = \int\left(\nabla \bullet \begin{bmatrix} x \\ 0 \end{bmatrix}\right) da = \oint x \hat{n}_x dl \qquad (12\text{-}14)$$

You need to compute the integral in the deformed geometry, which you accomplish by selecting the deformed frame when computing the integral. The software adds a deformed frame automatically when you add a contact pair.

MATERIAL PROPERTIES

- The rubber is hyperelastic and is modeled as a Mooney-Rivlin material with C_{10} = 0.37 MPa and C_{01} = 0.11 MPa. The material is almost incompressible, so the bulk modulus is set to 10^4 MPa, and the mixed formulation option is used.
- The compression of the confined air is assumed to be adiabatic, giving the pressure-density relation

$$\frac{p}{p_0} = \left(\frac{\rho}{\rho_0}\right)^{\gamma} = \left(\frac{A_0}{A}\right)^{\gamma} \qquad (12\text{-}15)$$

Here the cross-section area is denoted by A, with the undeformed value A_0 = 123.63 mm^2. The constant γ has the value 1.4 and p_0 = 0.1 MPa is the standard air pressure. The load acting on the interior of the seal is then

$$\Delta p = p - p_0 = p_0\left(\left(\frac{A_0}{A}\right)^{\gamma} - 1\right) \qquad (12\text{-}16)$$

CONSTRAINTS AND LOADS

- The lower straight part of the seal is glued to the car body, so all displacements are constrained there.
- One contact pair between the cylinder and the seal.
- One contact pair between the stationary plate and the seal.
- The rigid cylinder is lowered using the parameter of the parametric solver as the negative y displacement. It starts with a gap of 0 mm and is lowered 4 mm.

Results and Discussion

The deformed shape at the lowest cylinder position—corresponding to an indentation of 4 mm—without internal pressure is shown in Figure 12-4. The deformation scale is 1:1, that is, a true shape.

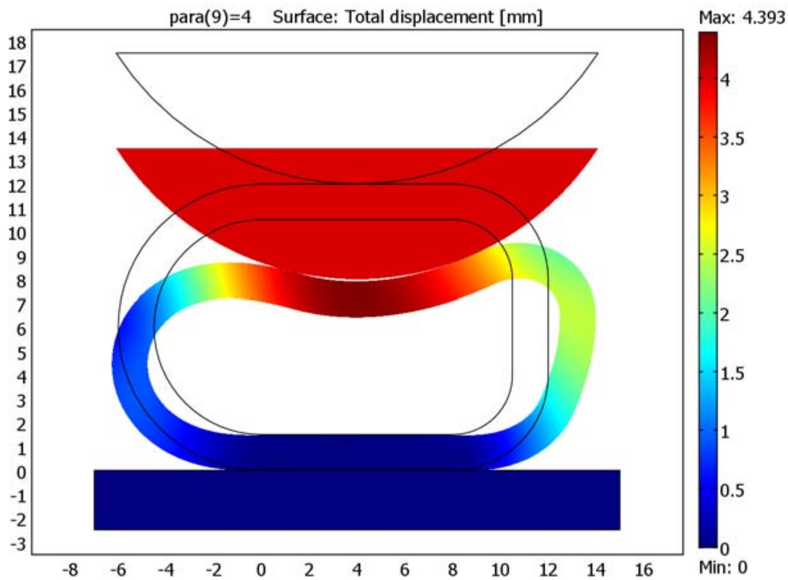


Figure 12-4: Seal deformation at 4 mm indentation with internal pressure neglected.

In Figure 12-5 you can compare the deformed shapes at 2 mm indentation. In the right figure, where the pressure is included, the seal profile appears "inflated."

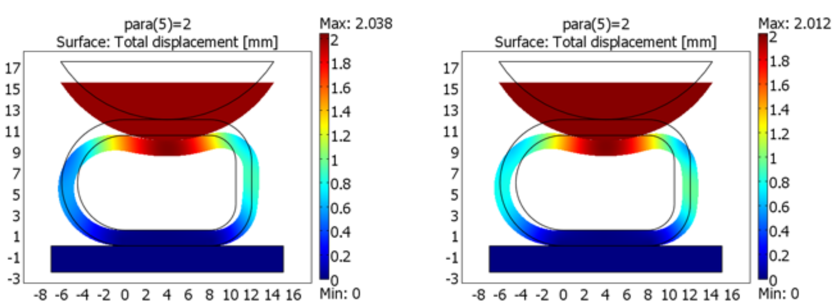


Figure 12-5: Seal deformation at 2 mm indentation without internal pressure (left) and with internal pressure (right).

Figure 12-6 contains a plot of the force (per unit length) versus compression (displacement of the rigid cylinder) with and without the internal pressure taken into account.

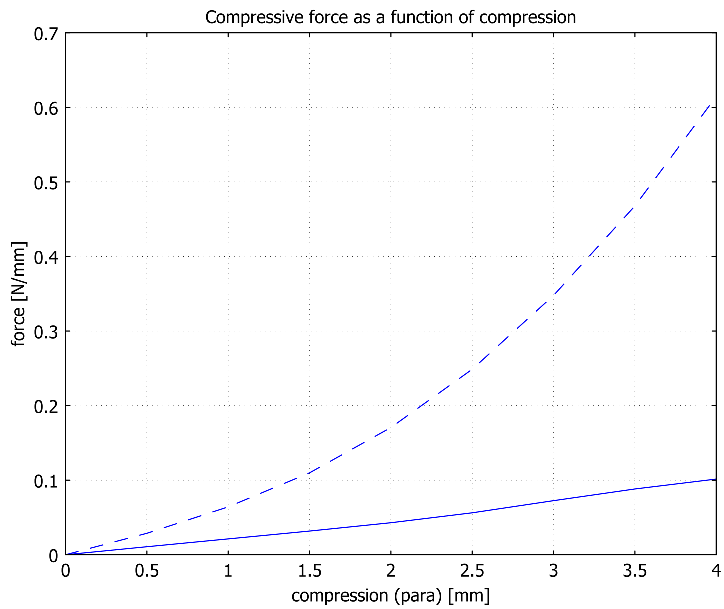


Figure 12-6: Force per unit length versus compression with (dashed) and without internal pressure.

Notice that the forces needed to compress the seal can be up to one order of magnitude larger when the effect of the air is taken into account.

In reality, a car door seal contains small holes through which the air can escape as long as the compression is not too fast. Thus the computed values are the limits corresponding to very slow and very fast compression, respectively.

Model Library path: `Structural_Mechanics_Module/Nonlinear_Material_Models/hyperelastic_seal`

MODEL NAVIGATOR

1 Select **2D** in the **Space dimension** list on the **New** page in the **Model Navigator**.

2 Select **Structural Mechanics Module>Plane Strain>Parametric analysis**.

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

OPTIONS AND SETTINGS

1 Select **Model Settings** from the **Physics** menu to open the **Model Settings** dialog box.

2 Select **MPa** from the **Base unit system** list to use mm as length scale unit and MPa as stress unit. Click **OK**.

3 Choose **Axes/Grid Settings** from the **Options** menu and give axis and grid settings according to the following table (first clear the **Auto** check box). When done, click **OK**.

AXIS		GRID	
x min	-7	x spacing	0.5
x max	15	Extra x	
y min	-3	y spacing	0.5
y max	25	Extra y	

4 Choose **Expressions>Global Expressions** from the **Options** menu and enter expression names, expressions, and descriptions (optional) according to the following table. When done, click **OK**.

NAME	EXPRESSION	DESCRIPTION	UNIT
int_p	0.1[MPa]*((123.63[mm^2]/int_area[mm^2])^1.4-1)	Internal pressure in cavity	MPa

GEOMETRY MODELING

1 Click the **Ellipse/Circle (Centered)** toolbar button and click the left mouse button at (0, 6). Move the mouse to (–6, 0) and click the left mouse button again.

2 Click the **Ellipse/Circle (Centered)** toolbar button and click the left mouse button at (8, 4). Move the mouse to (12, 0) and click the left mouse button again.

3 Click the **Ellipse/Circle (Centered)** toolbar button and click the left mouse button at (8, 8). Move the mouse to (12, 12) and click the left mouse button again.

4 Click the **Rectangle/Square** toolbar button and click the left mouse button at (0, 0). Move the mouse to (8, 12) and click the left mouse button again.

5 Click the **Rectangle/Square** toolbar button and click the left mouse button at (12, 4). Move the mouse to (7, 8) and click the left mouse button again.

6 Select all objects, and click the **Union** toolbar button.

7 Click the **Delete Interior Boundaries** toolbar button

8 Click the **Ellipse/Circle (Centered)** toolbar button and click the left mouse button at (0, 6). Move the mouse to (–4.5, 1.5) and click the left mouse button again.

9 Click the **Ellipse/Circle (Centered)** toolbar button and click the left mouse button at (8, 4). Move the mouse to (10.5, 1.5) and click the left mouse button again.

10 Click the **Ellipse/Circle (Centered)** toolbar button and click the left mouse button at (8, 8). Move the mouse to (10.5, 10.5) and click the left mouse button again.

11 Click the **Rectangle/Square** toolbar button and click the left mouse button at (0, 1.5). Move the mouse to (8, 10.5) and click the left mouse button again.

12 Click the **Rectangle/Square** toolbar button and click the left mouse button at (10.5, 4). Move the mouse to (7, 8) and click the left mouse button again.

13 Select all solid objects and click the **Difference** toolbar button.

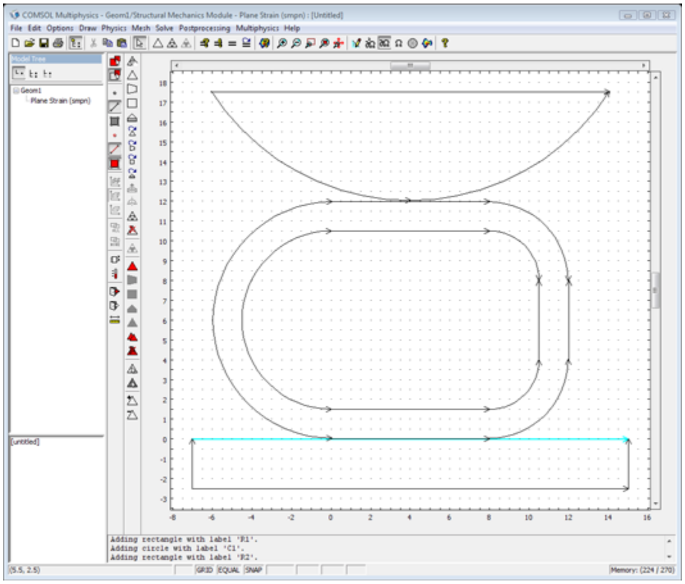
14 Click the **Rectangle/Square** toolbar button and click the left mouse button at (–7, –2.5). Move the mouse to (15, 0) and click the left mouse button again.

15 Click the **Ellipse/Circle (Centered)** toolbar button and click the left mouse button at (4, 24). Move the mouse to (16, 24) while pressing the Shift key and click the left mouse button again.

16 Click the **Rectangle/Square** toolbar button and click the left mouse button at (–7, 12). Move the mouse to (15, 17.5) and click the left mouse button again.

17 Select the upper circle and rectangle solid objects and click the **Intersection** toolbar button.

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



19 Select **Use Assembly** from the **Draw** menu to leave Draw mode using an assembly instead of a composite geometry.

PHYSICS SETTINGS

Contact Pairs

To model the contact, use contact pairs. Start by creating two contact pairs, one at the top and one at the bottom. Make the weaker boundary the slave and the stiffer one the master boundary.

1 Select **Physics>Contact Pairs** to open the **Contact Pairs** dialog box.
2 Click the **New** button to create a contact pair.
3 Select Boundary 6 and 7 in the **Master boundaries** list.
4 Click the **Check Selected** button below the **Master boundaries** list.
5 Select Boundary 11, 15, and 21 in the **Slave boundaries** list.
6 Click the **Check Selected** button below the **Slave boundaries** list.
7 Click the **New** button to create an additional contact pair.
8 Select Boundary 3 in the **Master boundaries** list.
9 Click the **Check Selected** button below the **Master boundaries** list.

10 Select Boundary 14 and 18 in the **Slave boundaries** list.

11 Click the **Check Selected** button below the **Slave boundaries** list.

12 Click **OK** to close the **Contact Pairs** dialog box.

Boundary Integration Variables

1 Select **Integration Coupling Variables>Boundary Variables** from the **Options** menu.

2 Select Boundaries 9, 10, 12, 16, 17, 19, and 20, then enter the data for the contour integral giving the internal area according to the following table:

NAME	EXPRESSION	FRAME
`int_area`	`-x2*nx2`	`Frame (deform)`

3 Select Boundaries 11, 15, and 21, then enter the data giving the total vertical force using the contact pressure from the first contact pair and the deformed frames normal direction according to the table below:

NAME	EXPRESSION	FRAME
`force`	`Tn_cp1_smpn*ny2`	`Frame (ref)`

Boundary Settings

Constrain the displacements of the glued lower part of the seal in both directions. Specify contact forces at the bottom and from the indenting cylinder.

Select **Boundary Settings** from the **Physics** menu. Specify boundary and pair settings according to the following tables:

	BOUNDARY 8	
Page	Constraint	
	Constraint condition	Fixed

	PAIRS 1, 2	
Page	Contact	
	pn	`20[MPa]/0.5[mm]*min(1e-3*5^auglagiter,10)`
Page	Contact, Initial	
	T_n	0

Subdomain Settings

1 Select **Subdomain Settings** from the **Physics** menu. Select Subdomain 3 then specify material parameters according to the following table:

SETTING	VALUE	UNIT
Material model	Hyperelastic	
Use mixed U-P formulation (nearly incompressible material)	√	
Hyperelastic model	Mooney-Rivlin	
C_{10}	`0.37`	MPa
C_{01}	`0.11`	MPa
κ	`1e4`	MPa
ρ	`1.1e-9`	t/mm^3

2 Select Subdomain 1, click the **Constraints** page, and select **Fixed** from the **Constraint condition** list.

3 Select Subdomain 2, click the **Constraints** page, and select **Prescribed displacement** from the **Constraint condition** list. Click the $\mathbf{R_x}$ and $\mathbf{R_y}$ check boxes and enter `-para` in the $\mathbf{R_y}$ edit field. Click **OK**.

MESH GENERATION

1 Select **Free Mesh Parameters** from the **Mesh** menu.

2 Click the **Subdomain** page, select Subdomain 3 in the **Subdomain selection** list. Enter `0.5` in the **Maximum element size** edit field.

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

4 Click the **Initialize Mesh** button on the Main toolbar to generate the mesh.

COMPUTING THE SOLUTION

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

2 Type the name `para` in the **Parameter name** edit field.

3 Type `0:0.5:4` in the **Parameter values** edit field.

4 Click the **Stationary** page.

5 Enter `0.01` in the **Tolerance** edit field in the **Augmented Lagrangian solver** area.

6 Enter `50` in the **Maximum number of iterations** edit field in both the **Augmented Lagrangian solver** area and the **Nonlinear settings** area.

7 Click the **Advanced** page and select **Manual** in the **Type of scaling** list.

8 Enter `u 5 v 5 p 0.5 Tn_cp1_smpn 0.2 Tn_cp2_smpn 0.2` in the **Manual scaling** edit field.

Because the initial value of the contact pressure is zero, you need to use manual scaling of the variables. Read more about scaling of variables for contact problems in "Solver Settings for Contact Modeling" on page 129 of the *Structural Mechanics Module User's Guide*.

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

10 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To reproduce the plot in Figure 12-4 execute the following instructions.

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

2 On the **General** page keep the default parameter value in the **Solution to use** area. (This corresponds to the last solution that was calculated; to generate the left plot in Figure 12-5 instead, choose **2** from the **Parameter value** list.)

3 Click the **Surface** tab. From the **Predefined quantities** list select **Total displacement**.

4 Click **OK** to close the dialog box and generate the plot.

Next, turn to the plots of force versus displacement displayed in Figure 12-6. The solid line, which corresponds to the case of neglecting any internal pressure inside the seal, is obtained as follows:

1 Select **Global Variables Plot** from the **Postprocessing** menu.

2 Enter force in the **Expression** edit field.

3 Click the > button to move force to the **Quantities to plot** list.

4 Select all parameter values from 0 to 4 in the **Solutions to use** list.

5 Click **OK** to plot the compressive force and close the dialog box.

The plot appears in a separate figure window; keep this window to allow for a direct comparison with the case of an airtight seal.

ADDING INTERNAL PRESSURE

1 Select **Boundary Settings** from the **Physics** menu, then click the **Load** tab.

2 Select Boundaries 9, 10, 12, 16, 17, 19, and 20.

3 Select **Follower type** from the **Type of load** list, then set **P** to `int_p`.

4 Click **OK**.

COMPUTING THE SOLUTION

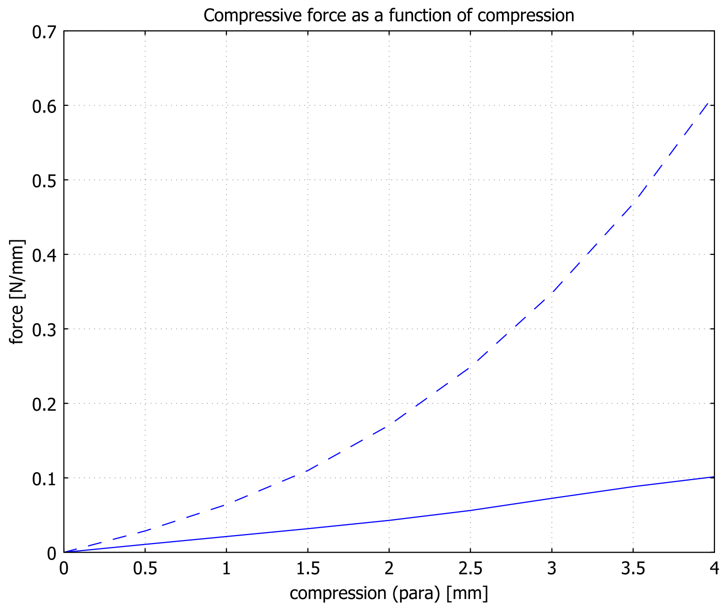
Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

Plot the force versus displacement curve for an airtight seal by following the steps below.

1. Select **Global Variables Plot** from the **Postprocessing** menu.
2. Click the **Title/Axis** button. Click the option button next to the **Title** edit field and then type `Compressive force as a function of compression`.
3. Click the option button next to the **First axis label** edit field and then type `compression (para) [mm]`. Click the option button next to the **Second axis label** edit field and type `force [N/mm]`. Click **OK**.
4. Enter force in the **Expression** edit field.
5. Click the > button to move force to the **Quantities to plot** list.
6. Select all parameter values from 0 to 4 in the **Solutions to use** list.
7. Select the **Keep current plot** check box.
8. Select **Figure 1** in the **Plot in** list.
9. Click the **Line Settings** button to open the **Line Settings** dialog box.
10. Select **Dashed line** in the **Line style** list and click **OK** to close the **Line Settings** dialog box.
11. Click **OK** to plot the compressive force and close the dialog box.

The figure window now contains the plots in Figure 12-6 on page 470, which is repeated below for convenience.



Thermally Induced Creep

Introduction

This model computes the stress history over a long time for a material that exhibits creep behavior. The model is taken from *NAFEMS Understanding Non-Linear Finite Analysis Through Illustrative Benchmarks* (Ref. 1). The displacement and stress levels are compared with the values given in the reference.

Creep is an inelastic time-dependent deformation that occurs when a material is subjected to stress (typically much less than the yield stress) at sufficiently high temperature, say 40% of the melting point or more.

The creep strain rate, in a general case, depends on stress, temperature, and time, usually in a nonlinear manner:

$$\dot{\varepsilon}_c = \dot{\varepsilon}_c(\sigma, T, t)$$

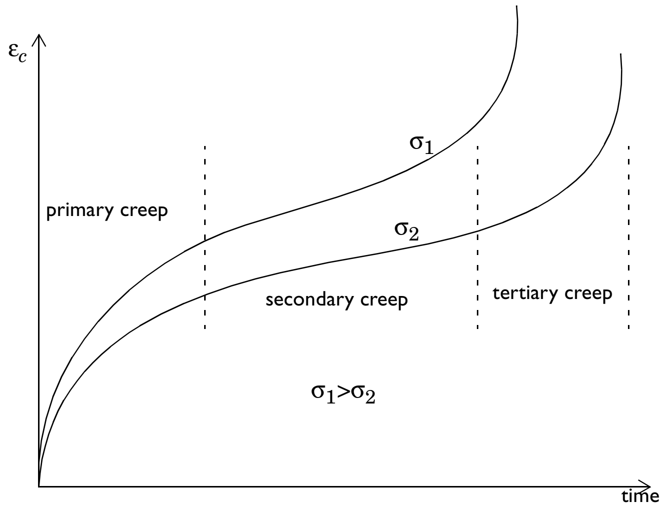
Fortunately, it is usually possible to separate these effects in the way indicated by the following equation:

$$\dot{\varepsilon}_c = f_1(\sigma)f_2(T)f_3(t)$$

Experimental creep data (using constant stress and temperature) often display three different types of behavior for the creep strain rate as function of time:

- In the initial *primary creep* regime the creep strain rate is decreasing with time.

- In the *secondary creep* regime the creep strain rate is almost constant.
- In the final *tertiary creep* regime the creep strain rate is increasing with time until a failure occurs.



Most of the time is spent in secondary creep. Tertiary creep is seldom important because it only accounts for a small fraction of the total lifetime, just before failure.

The most common model for secondary creep is the *Norton equation* where the creep strain rate is proportional to a power of the stress:

$$\dot{\varepsilon}_{\mathrm{c}} = A\sigma^n f_2(T)$$

In the multiaxial generalization of Norton's law, the creep strain rate tensor is proportional to the stress deviator, s:

$$\dot{\varepsilon}_{\mathrm{c}} = \frac{3}{2\tau}\left(\frac{\sigma_{\mathrm{e}}}{\sigma_{\mathrm{c}}}\right)^{n-1}\frac{s}{\sigma_{\mathrm{c}}}f_2(T)$$

Here σ_{e} is the von Mises effective stress, and the time constant, τ, and the creep stress, σ_{c}, have been introduced to avoid material data with strange units. When a uniaxial test is made using the stress σ_{c}, the creep strain rate will be τ^{-1}. If, as is common, creep data is supplied as "50 MPa will give 1% strain in 10000 h," it is possible to identify $\sigma_{\mathrm{c}} = 50$ MPa and $\tau = 10^6$ h.

The different sets of constants are related through

$$A = \frac{1}{\tau(\sigma_c)^n}$$

giving the following expression for σ_c where τ can be given arbitrarily if A is known.

$$\sigma_c = (A\tau)^{-\frac{1}{n}}$$

In primary creep, the relation can be augmented by an explicit dependence on the time such as

$$\dot{\varepsilon}_c = A\sigma^n m t^{m-1} f_2(T)$$

In this equation, $0 < m < 1$ so that the strain rate is decreasing with time. It is infinite at time 0, so you must be careful when using this relation in a numerical method.

When the complete process with both primary and secondary creep is simulated, it is possible to combine the models as in the equation

$$\dot{\varepsilon}_c = (A_1\sigma^{n_1} + A_2\sigma^{n_2} m t^{m-1}) f_2(T)$$

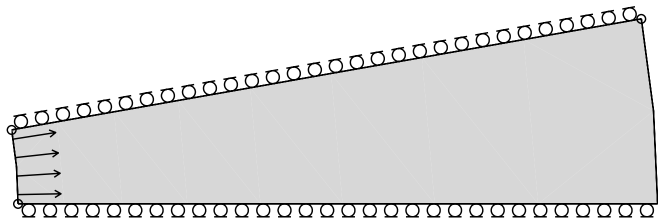
A creep model assumes that the total strain is the sum of the elastic strain, the creep strain, and possibly a thermal strain. The stresses are computed from the elastic strain as for other materials.

In the Structural Mechanics Module you can introduce the creep strains as new dependent variables. Because the creep strains are subtracted from the total strains during the stress computations, you should select shape functions for the creep strains which are one order lower than what is used for the displacements.

Model Definition

The model is called “Thermally Induced Creep Benchmark” and is described in detail in section 3.8 of *NAFEMS Understanding Non-Linear Finite Analysis Through Illustrative Benchmarks* (Ref. 1).

The geometry is a hollow sphere with inner radius 200 mm and outer radius 500 mm. The problem is axisymmetric. Actually, the solution depends only on the radial coordinate, so you could select any section having radial cuts as the computational domain. To follow the original example the model uses a 10-degree sector.



MATERIAL PROPERTIES

- Isotropic with $E = 10\cdot 10^3$ MPa, $\nu = 0.25$.
- Secondary Creep Data:

$$\dot{\varepsilon}_c = A_1\sigma^{n_1}f_2(T)$$

with $A_1 = 3.0\cdot 10^{-6}$ (t in hours, σ in MPa), $n_1 = 5.5$, and $f_2(T) = e^{-12500/T}$ (T in K).

- Primary Creep Data (only used in a modified version of the problem):

$$\dot{\varepsilon}_c = A_2\sigma^{n_2}mt^{m-1}f_2(T)$$

$A_2 = 10$ (t in hours, σ in MPa), $n_2 = 3.5$, and $m = 0.5$.

LOADS

- An internal pressure of 30 MPa. Strictly speaking, the load should be applied in an elastic step to provide initial conditions for the creep analysis. It is more practical, however, to apply the load fast in the beginning of the creep analysis. The load thus grows from zero to full value in the first 0.1 s of this analysis.
- A temperature field with the distribution $T = 333(1 + 100/r)$, where r and z are given in millimeters and T in kelvin. In COMSOL Multiphysics, you enter this as `333[K]*(1+100[mm]/sqrt(r*r+z*z))`.

CONSTRAINTS

Symmetry constraint conditions on the radial edges.

Results and Discussion

The reference solutions are available only in graphical form, and the corresponding COMSOL Multiphysics results appear in the following plots. The correspondence is good in all respects.

An interesting feature of this problem is the extreme variation in the time scales over which different phenomena occur. The time marching schemes in COMSOL Multiphysics handle this automatically.

Figure 12-7 shows the von Mises effective stress at 10^8 h.

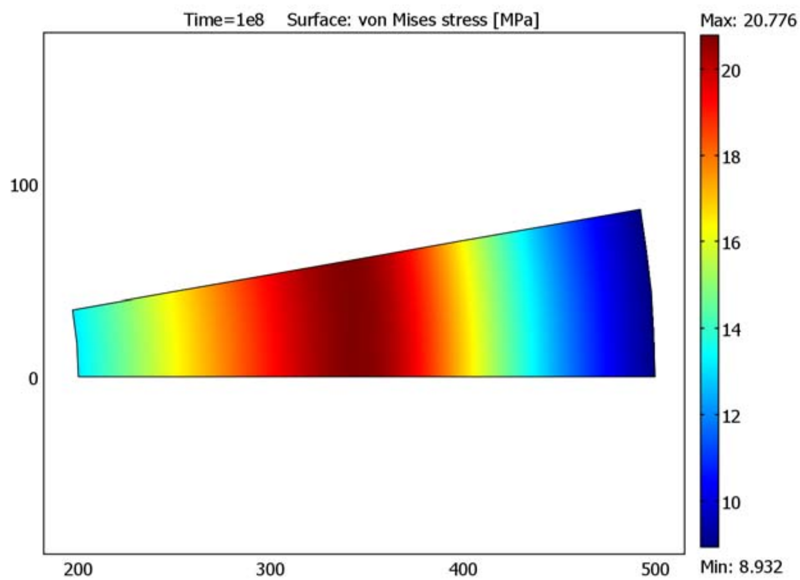


Figure 12-7: Von Mises effective stress at $t = 10^8$.

The evolution of displacement with time appears in Figure 12-8. The upper curve represents the inner radius, and the lower curve represents the outer radius. In the following table you can compare the values at time 10^{10} h with the reference values.

LOCATION	COMSOL MULTIPHYSICS	REFERENCE (REF. 1)
r=200	26.1 mm	25.3 ± 0.7 mm
r=500	4.2 mm	4.0 ± 0.7 mm

Because the reference values are given as graphs, these values include an estimate of the error caused by reading this graph.

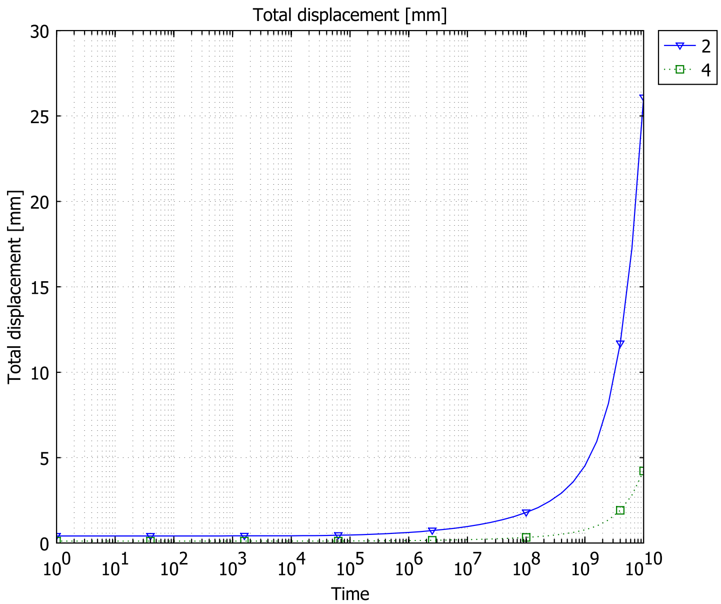


Figure 12-8: Radial displacement at r = 200 *mm (Point 2) and* 500 *mm (Point 4) against* t.

The next graph shows the variation of the von Mises effective stress with time at the inner, middle, and outer radii. Notice that significant changes in the stress state occur already at the time 10^4 h, that is, after one millionth of the total analysis time. In the final state the stresses are completely redistributed. The creep process is much faster at the inside due to the temperature variation, so the stresses tend to relax there. The following table shows the final values of the von Mises effective stress at $t = 10^{10}$ h and the reference values for comparison.

LOCATION	**COMSOL MULTIPHYSICS**	**REFERENCE (REF. 1)**
r=205	11.5 MPa	11.3 ± 0.4 MPa
r=350	17.6 MPa	17.4 ± 0.4 MPa
r=495	21.1 MPa	21.1 ± 0.4 MPa

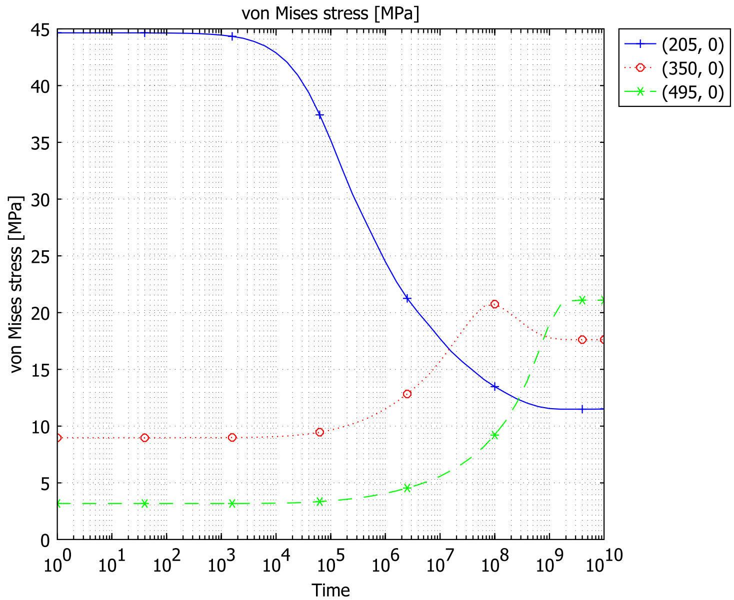


Figure 12-9: Von Mises effective stress at $r = 205$ *mm* (+), 350 *mm* (o), *and* 495 *mm* (*) *versus* t.

In the last graph, Figure 12-10, the von Mises effective stress distribution through the thickness is shown at the times 1, 10^4, 10^5, 10^7, 10^8, and 10^{10} (all times in hours). The stress at the inner radius decreases monotonically with time, but at other locations the state is more complex.

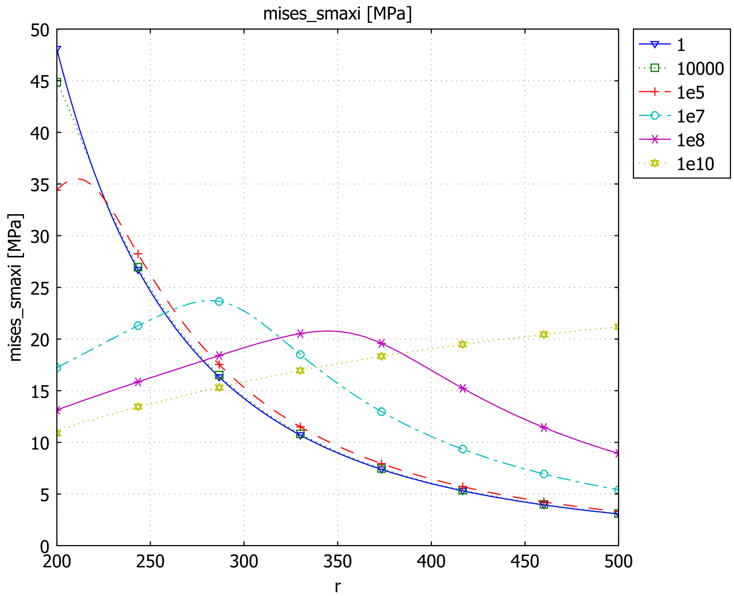


Figure 12-10: Von Mises effective stress versus r *at* $t = 1, 10^4, 10^5, 10^7, 10^8$, *and* 10^{10} *hours. Times increasing with decreasing stress at* $r = 200$ *mm.*

When you add primary creep the stresses are redistributed during the process, especially in the beginning where the contribution from the initial creep is large. The final radial displacement at the inner radius increases from 26.1 mm to 29.1 mm. The following three graphs show the results with primary creep included.

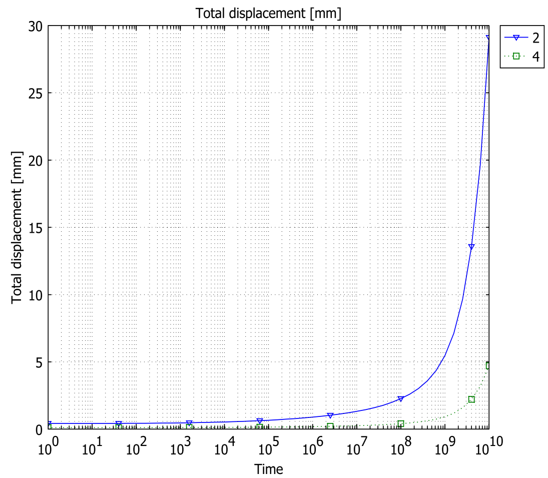


Figure 12-11: Radial displacement at $r = 200$ *mm* (point 2) *and* 500 *mm* (point 4) *versus* t*, primary creep included.*

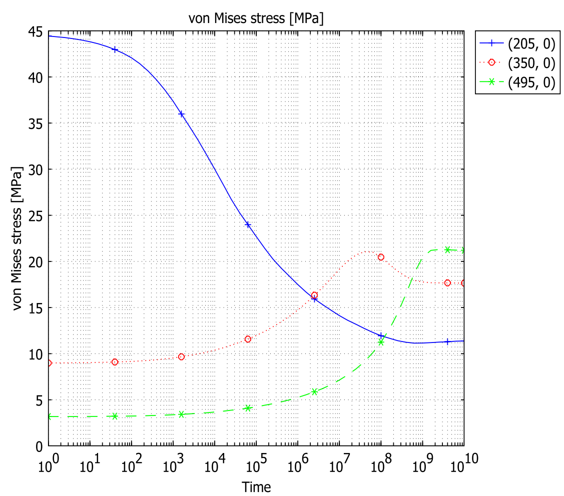


Figure 12-12: Von Mises effective stress at $r = 205$ *mm* (+), 350 *mm* (o), *and* 495 *mm* (*) *versus* t*, primary creep included.*

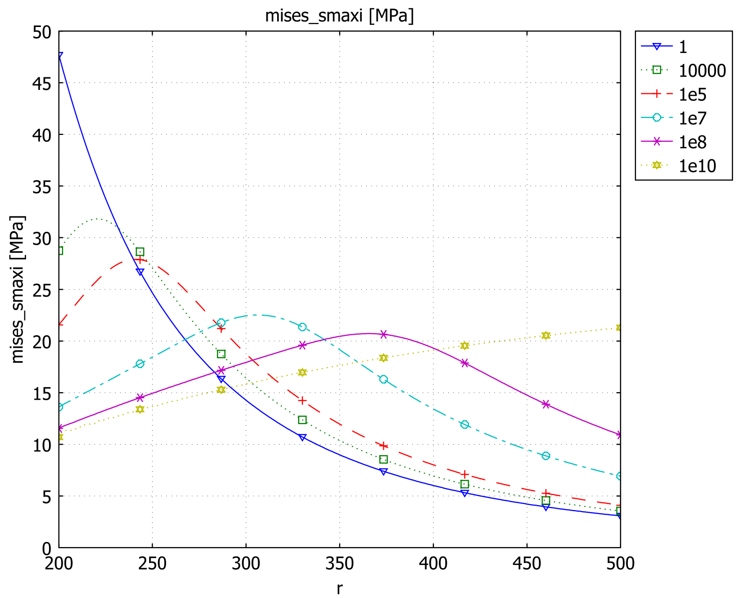


Figure 12-13: Von Mises effective stress versus r at t = 1, 10^4, 10^5, 10^7, 10^8, and 10^{10} hours, primary creep included. Times increasing with decreasing stress at r = 200 mm.

Reference

1. A.A. Becker, *Understanding Non-Linear Finite Analysis Through Illustrative Benchmarks, NAFEMS*, Glasgow, 2001.

Model Library path: `Structural_Mechanics_Module/Nonlinear_Material_Models/thermally_induced_creep`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **Axial symmetry (2D)** from the **Space dimension** list on the **New** page in the **Model Navigator**.

2 Select **Structural Mechanics Module>Axial Symmetry, Stress-Strain>Quasi-static analysis**.

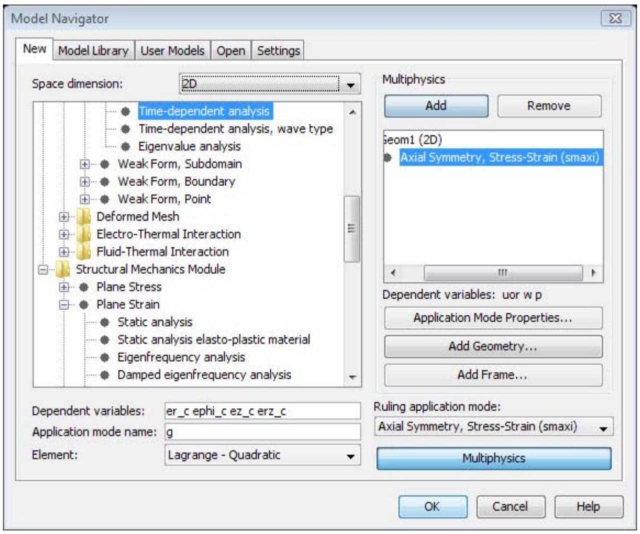
3 Select **Lagrange - Cubic** from the **Element** list.

4 Click **Multiphysics** and then click **Add**.

5 Select **2D** from the **Space dimension** list.

6 Select **COMSOL Multiphysics>PDE Modes>PDE, General Form>Time-dependent analysis**.

7 In the **Dependent variables** edit field type `er_c ephi_c ez_c erz_c`.



8 In the **Application mode name** edit field, type `creep`.

9 Click the **Add** button, and then click **OK**.

OPTIONS AND SETTINGS

1 Select **Model Settings** from the **Physics** menu to open the **Model Settings** dialog box.

2 Select **MPa** from the **Base unit system** list to use mm as length unit and MPa as stress unit.

3 Select **Axes/Grid Settings** from the **Options** menu and give axis and grid settings according to the following table:

AXIS		GRID	
r min	-600	r spacing	100
r max	600	Extra r	

AXIS		GRID	
z min	-600	z spacing	100
z max	600	Extra z	

4 Select **Constants** from the **Options** menu and enter constant names, expressions, and descriptions (optional) according to the following table:

NAME	EXPRESSION	DESCRIPTION	UNITS
A2	10	Coefficient for primary creep	
n2	3.5	Stress exponent for primary creep	
m	0.5	Time exponent for primary creep	
A1	3e-6	Coefficient for secondary creep	
n1	5.5	Stress exponent for secondary creep	
tau	1	Creep time constant (arbitrary)	h
sig_cp	(A2*tau^m)^(-1/n2)	Primary creep reference stress	MPa
sig_cs	(A1*tau)^(-1/n1)	Secondary creep reference stress	MPa
p_const	3/2/tau/(sig_cp)^n2*m	Useful constant, primary creep	
s_const	3/2/tau/(sig_cs)^n1	Useful constant, secondary creep	

5 Select **Expressions>Scalar Expressions** from the **Options** menu and enter names, expressions, and descriptions (optional) for the scalar expression variables according to the following table:

NAME	EXPRESSION	DESCRIPTION	UNIT
sd_r	(2*sr_smaxi-sphi_smaxi-sz_smaxi)/3	Stress deviator r-component	MPa
sd_phi	(2*sphi_smaxi-sr_smaxi-sz_smaxi)/3	Stress deviator phi-component	MPa
sd_z	(2*sz_smaxi-sr_smaxi-sphi_smaxi)/3	Stress deviator z-component	MPa
T	333[K]*(1+100[mm]/sqrt(r*r+z*z))	Prescribed temperature field	K
f_t	exp(-12500[K]/T)	Temperature dependence of creep rate	
rate_p	p_const*f_t*mises2^((n2-1)/2)*(t)^(m-1)	Primary creep strain rate factor. Note small number to avoid t=0 problems.	
rate_s	s_const*f_t*mises2^((n1-1)/2)	Secondary creep strain rate factor	

NAME	EXPRESSION	DESCRIPTION	UNIT
rate_fac	rate_s	Total creep strain rate factor; only secondary creep in this analysis	
mises2	sr_smaxi^2+sphi_smaxi^2+sz_smaxi^2 -sr_smaxi*sphi_smaxi- sphi_smaxi*sz_smaxi- sr_smaxi*sz_smaxi+3*srz_smaxi^2	Effective stress squared	MPa^2

GEOMETRY MODELING

1 Click the **Ellipse/Circle (Centered)** toolbar button and click the left mouse button at (0, 0). Move the mouse to (500, 500) and click the left mouse button again to create a circle with the outer radius.

2 Click the **Ellipse/Circle (Centered)** toolbar button and click the left mouse button at (0, 0). Move the mouse to (200, 200) and click the left mouse button again to create a circle with the outer radius.

3 Select both circles, and click the **Difference** toolbar button.

4 Click the **Line** toolbar button and click the left mouse button at (0, 0). Move the mouse to (600, 0), click the left mouse button and click the right mouse button.

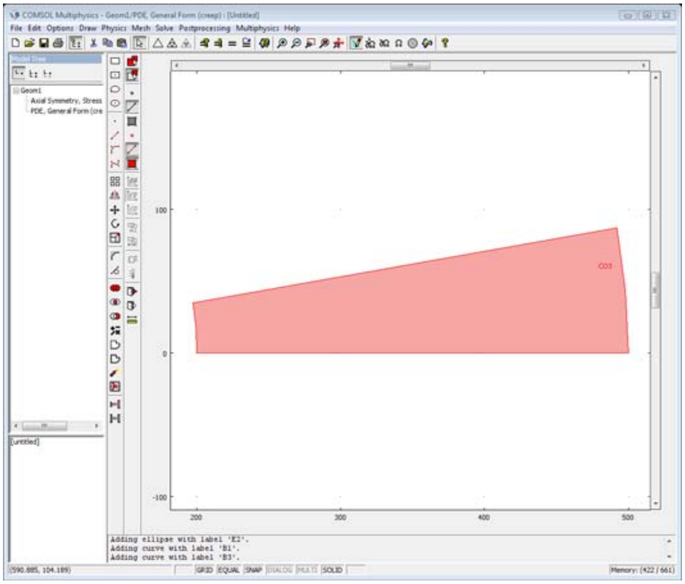
5 Make a copy of the line, and click the **Rotate** toolbar button. Type `10` in the **Rotation angle** edit field and click **OK**.

6 Click the **Line** toolbar button and click the left mouse button at (600, 0). Move the mouse so that the cursor snaps to the end of line B2, click the left mouse button and click the right mouse button.

7 Select the three lines and click the **Coerce to Solid** toolbar button.

8 Select both solid objects and click the **Intersection** toolbar button.

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



PHYSICS SETTINGS

Boundary Settings

Constrain the displacements of the cuts through the sphere to have symmetry conditions. Specify the pressure on the inner surface as increasing from zero to 30 MPa, over the time 0.1 s.

1 Select **Axial Symmetry, Stress-Strain (smaxi)** from the **Multiphysics** menu.

2 Select **Boundary Settings** from the **Physics** menu.

3 Specify boundary settings according to the following tables:

	BOUNDARIES 1, 2	
Page	Constraint	
	Constraint condition	Symmetry plane

	BOUNDARY 3	
Page	Load	
	Coordinate system	Tangent and normal coord. sys. (t, n)

	BOUNDARY 3	
	F_t	0
	F_n	-30[MPa]*((t>0.1[s])+t/(0.1[s])*(t<=0.1[s]))

4 Select **PDE, General Form (creep)** from the **Multiphysics** menu.

5 Select **Boundary Settings** from the **Physics** menu.

6 Specify boundary settings according to the following table:

	BOUNDARIES 1–4	
Page	Type	
	Boundary condition type	Neumann boundary condition

Subdomain Settings

1 Select **Axial Symmetry, Stress-Strain (smaxi)** from the **Multiphysics** menu.

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

3 Specify the material properties of the sphere according to the following table:

	SUBDOMAIN 1	
Page	Material	
	Material model	Isotropic
	E	10e3
	ν	0.25

4 Select **PDE, General Form (creep)** from the **Multiphysics** menu.

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

6 Specify the terms of the PDE according to the following table:

	SUBDOMAIN 1	
Page	Γ	
	Flux vector	0 0 0 0 0 0 0 0
Page	F	
	Source term	rate_fac*sd_r rate_fac*sd_phi rate_fac*sd_z rate_fac*srz_smaxi

	SUBDOMAIN 1	
Page	d_a	
	Damping/Mass coefficient	1 0 0 0 0 1 0 0 0 0 1 0 0 0 0 1

7 Select **Equation System>Subdomain Settings** from the **Physics** menu.

8 Change the definitions of the stresses, so that each occurrence of the total strain is replaced by the difference between the total strain and the creep strain. The additional terms are in italics in the table below.

	SUBDOMAIN 1	
Page	Variables	
	sr_smaxi	E_smaxi*(1-nu_smaxi)*(er_smaxi-*er_c*)/((1+nu_smaxi)*(1-2*nu_smaxi))+E_smaxi*nu_smaxi*(ephi_smaxi-*ephi_c*)/((1+nu_smaxi)*(1-2*nu_smaxi))+E_smaxi*nu_smaxi*(ez_smaxi-*ez_c*)/((1+nu_smaxi)*(1-2*nu_smaxi))
	sz_smaxi	E_smaxi*nu_smaxi*(er_smaxi-*er_c*)/((1+nu_smaxi)*(1-2*nu_smaxi))+E_smaxi*nu_smaxi*(ephi_smaxi-*ephi_c*)/((1+nu_smaxi)*(1-2*nu_smaxi))+E_smaxi*(1-nu_smaxi)*(ez_smaxi-*ez_c*)/((1+nu_smaxi)*(1-2*nu_smaxi))
	sphi_smaxi	E_smaxi*nu_smaxi*(er_smaxi-*er_c*)/((1+nu_smaxi)*(1-2*nu_smaxi))+E_smaxi*(1-nu_smaxi)*(ephi_smaxi-*ephi_c*)/((1+nu_smaxi)*(1-2*nu_smaxi))+E_smaxi*nu_smaxi*(ez_smaxi-*ez_c*)/((1+nu_smaxi)*(1-2*nu_smaxi))
	srz_smaxi	E_smaxi*(erz_smaxi-*erz_c*)/(1+nu_smaxi)

MESH GENERATION

1 Select **Free Mesh Parameters** from the **Mesh** menu.

2 Select **Finer** in the **Predefined mesh sizes** list and click the **Remesh** button.

3 Click **OK**.

4 Click the **Zoom Extents** button on the Main toolbar to examine the mesh.

COMPUTING THE SOLUTION

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

2 Specify the solver parameters according to the following table; then click **OK**:

Page	General	
	Times	`[0 logspace(-2,10,61)]`
	Absolute tolerance	`1e-6`

The tighter absolute tolerance provides increased accuracy during the time stepping.

3 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

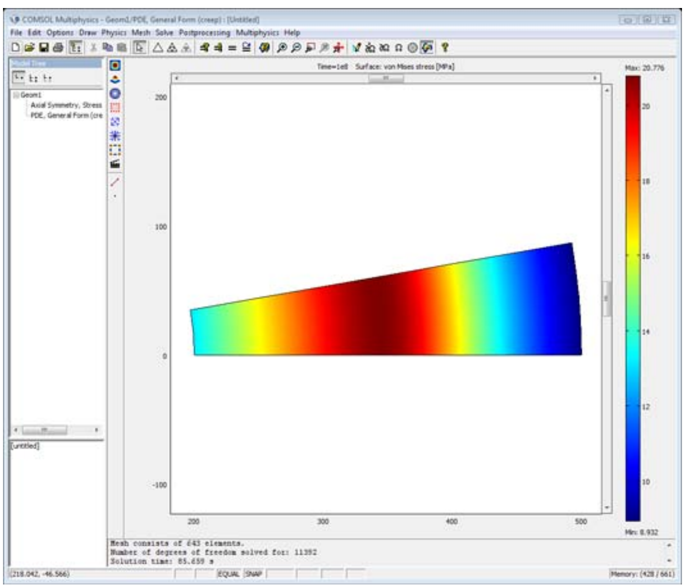
Plot the von Mises effective stress at 10^8 h.

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

2 Select **1e8** from the **Solution at time** list on the **General** page.

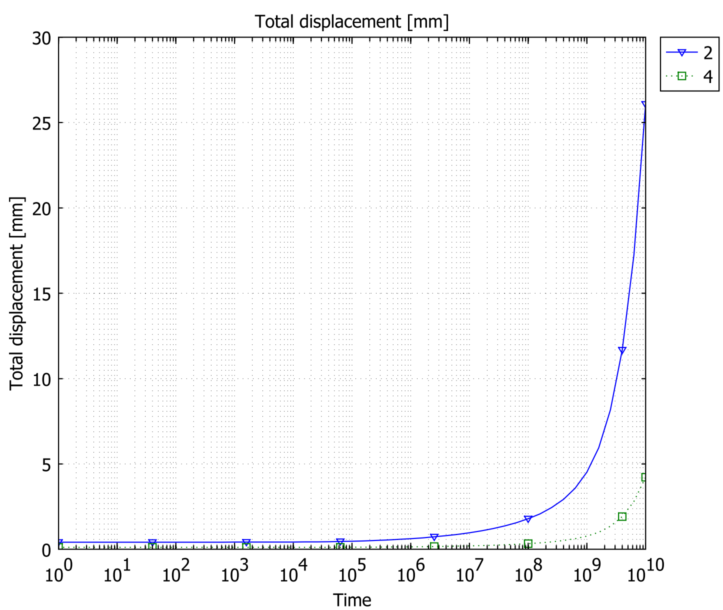
3 Select **von Mises stress** from the **Predefined quantities** list on the **Surface** page (already selected as it is the default plot).

4 Click **OK** to close the **Plot Parameters** dialog box and plot the von Mises stress.



Plot the radial displacement at the inner and outer radii versus t.

1 Select **Domain Plot Parameters** from the **Postprocessing** menu.
2 Select all times between **1** and **1e10** in the **Solutions to use** list on the **General** page.
3 Click the **Title/Axis** button.
4 Click the **Log scale** check box for the **First axis label**.
5 Click **OK** to close the **Title/Axis Settings** dialog box.
6 Click the **Point** tab.
7 From the **Predefined quantities** list in the **y-axis data** area select **Axial Symmetry, Stress-Strain (smaxi)>r-displacement**.
8 Select Points 2 and 4 in the **Point selection** list to get the radial displacement at the inner radius and outer radius.
9 Click the **Line Settings** button.
10 Click the **Legend** check box in the **Line Settings** dialog box.
11 Click **OK** to close the **Line Settings** dialog box.
12 Click **OK** to close the **Domain Plot Parameters** dialog box and plot the radial displacement.



Plot the von Mises effective stress versus r for different times.

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

2 Select `1`, `10000`, `1e5`, `1e7`, `1e8`, and `1e10` as times to plot in the **Solutions to use** list on the **General** page.

3 Click the **Title/Axis** button.

4 Click the **Log scale** check box for the **First axis label** to disable the logarithmic scale on the *x*-axis.

5 Click **OK** to close the **Title/Axis Settings** dialog box.

6 Click on the **Line/Extrusion** page.

7 Select Boundary 2 in the **Boundary selection** list.

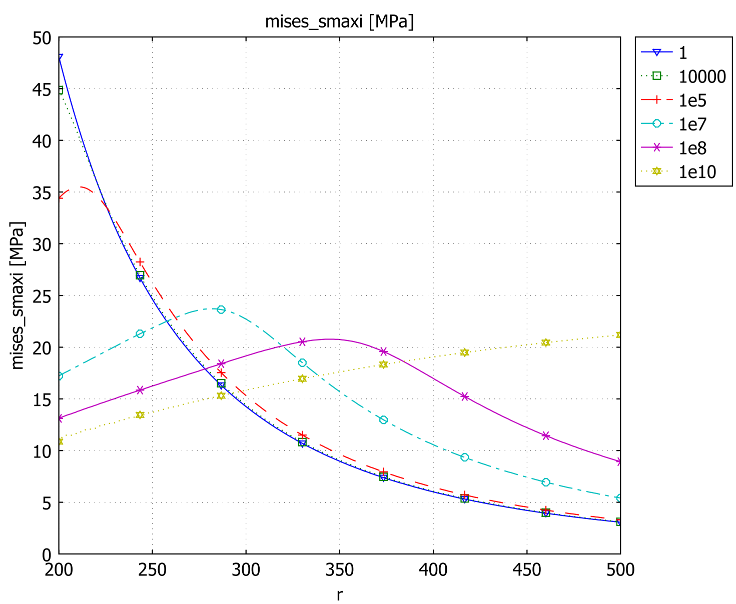
8 Enter `mises_smaxi` in the **Expression** edit field.

9 Click the upper option button in the **x-axis data** area, then select **r** from the associated list.

10 Click the **Line Settings** button.

11 Click the **Legend** check box in the **Line Settings** dialog box; then click **OK**.

12 Click **OK** to plot the von Mises effective stress against the radius.



ENABLING PRIMARY CREEP

Select **Expressions>Scalar Expressions** from the **Options** menu and change the expression for the creep strain rate factor to include primary creep:

NAME	EXPRESSION	DESCRIPTION
`rate_fac`	`rate_s+rate_p`	Total creep strain rate factor

COMPUTING THE SOLUTION

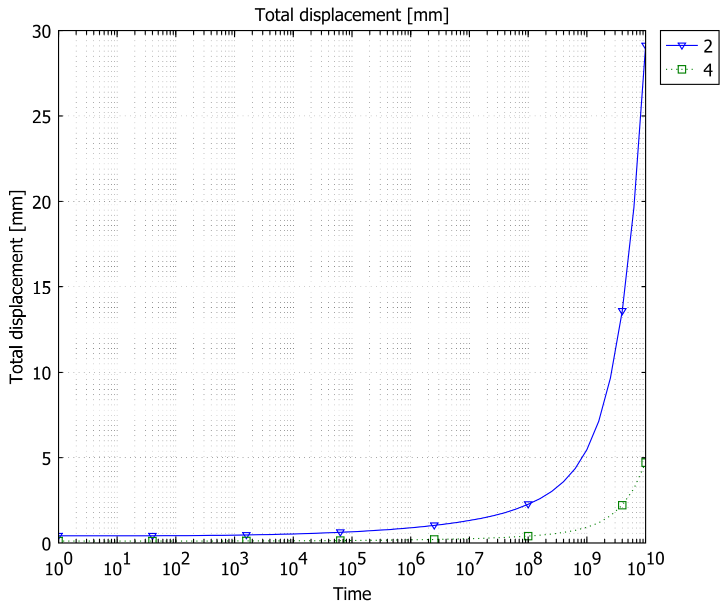
Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

Plot the radial displacement at the inner and outer radii versus *t*:

1. Select **Domain Plot Parameters** from the **Postprocessing** menu.
2. Select all times between **1** and **1e10** in the **Solutions to use** list on the **General** page.
3. Click the **Title/Axis** button.
4. Click the **Log scale** check box for the **First axis label**.
5. Click **OK** to close the **Title/Axis Settings** dialog box.
6. Click the **Point** tab.
7. From the **Predefined quantities** list in the **y-axis data** area select **Axial Symmetry, Stress-Strain (smaxi)>r-displacement**.
8. Select Points 2 and 4 in the **Point selection** list to get the radial displacement at the inner radius and outer radius.
9. Click the **Line Settings** button.
10. Click the **Legend** check box in the **Line Settings** dialog box.
11. Click **OK** to close the **Line Settings** dialog box.

12 Click **OK** to close the **Domain Plot Parameters** dialog box and plot the radial displacement.



Stresses in the Soil Surrounding a Traffic Tunnel

Introduction

This model demonstrates the use of elastoplastic analysis in soil mechanics using the Mohr-Coulomb material model under plane strain conditions. The analysis studies the stresses and deformations in the soil surrounding a traffic tunnel. This model is inspired by a model in Ref. 1.

Model Definition

The modeling of the tunnel and the surrounding soil consist of the following stages:

1. Excavation of the soil at the site.
2. Adding the tunnel onto the soil and applying boundary loads on the tunnel.

Due to symmetry reason the model only includes one half of the geometry (see Figure 12-14).

A suitable 2D application mode to model geotechnical problems as this is the Plane Strain application mode.

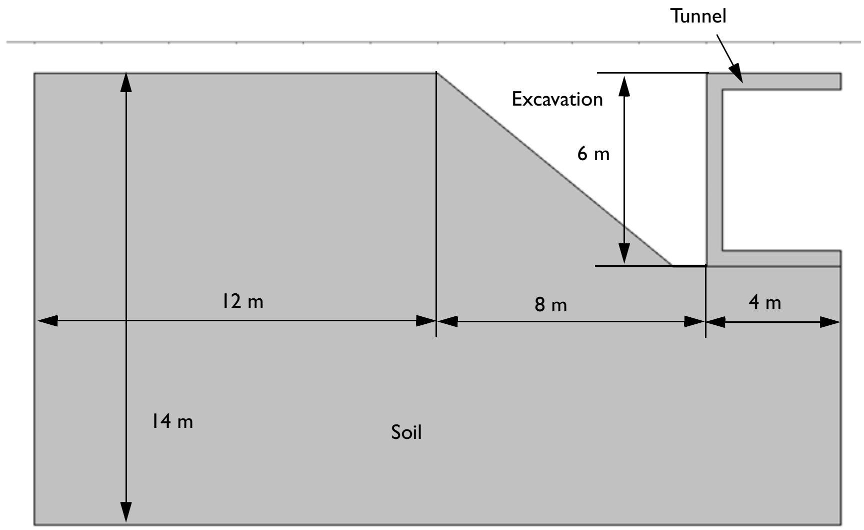


Figure 12-14: Cross section of the tunnel and the soil layer.

MATERIAL

Soil is a material with a highly nonlinear stress-strain behavior. The soil properties can be divided into strength and stiffness properties. The strength of soil can be characterized by the effective strength parameters c (cohesion) and ϕ (friction angle). The stiffness parameters E (Young's modulus) and ν (Poisson's ratio) describes the elastic deformation of the soil.

Problems of material failure in soil mechanics and other frictional materials such as concrete are often modeled using the well-known Mohr-Coulomb law. This is an elastic perfectly-plastic material model. A smooth approximation of the Mohr-Coulomb yield surface for a three-dimensional case is the Drucker-Prager model.

The failure criteria, the yield surface, for both material models are based on the two strength parameters, cohesion c and friction angle ϕ, and two stress quantities, the hydrostatic stress and the equivalent deviatoric stress.

The yield surface F for the Drucker-Prager material law is given by:

$$F = 3 \cdot \alpha \cdot \sigma_{\mathrm{m}} + \sigma_{\mathrm{eqv}} - K$$

where

$$\alpha = \frac{2 \cdot \sin\phi}{\sqrt{3} \cdot (3 - \sin\phi)}$$

and

$$K = \frac{6 \cdot c \cdot \cos\phi}{\sqrt{3} \cdot (3 - \sin\phi)}$$

where σ_m is the hydrostatic stress (or mean stress), and σ_eqv is the equivalent deviatoric stress.

On the other hand, if two-dimensional plane strain conditions prevails the Drucker-Prager criterion becomes identical to the Mohr-Coulomb criterion if you write the above material parameters as (see Ref. 3):

$$\alpha = \frac{\tan\phi}{\sqrt{(9 + 12 \cdot \tan^2\phi)}}$$

$$K = \frac{3 \cdot c}{\sqrt{(9 + 12 \cdot \tan^2\phi)}}$$

The hydrostatic stress is defined using the normal stress components σ_{ii} by (Ref. 2):

$$\sigma_\mathrm{m} = \frac{\sigma_{ii}}{3} = \frac{\sigma_x + \sigma_y + \sigma_z}{3}$$

whereas the equivalent deviatoric stress is defined using the deviatoric stress components s_{ij}:

$$\sigma_\mathrm{eqv} = \sqrt{\frac{1}{2} \cdot s_{ij} \cdot s_{ij}} = \sqrt{\frac{1}{2} \cdot (s_x^2 + s_y^2 + s_z^2) + s_{xy}^2 + s_{yz}^2 + s_{zx}^2}$$

where the deviatoric stress components s_{ij} are defined by:

$$s_{ij} = \sigma_{ij} - \frac{\delta_{ij} \cdot \sigma_{kk}}{3} \qquad \text{or}$$

$$\begin{aligned} s_x &= \sigma_x - \sigma_\mathrm{m} \qquad & s_{xy} &= \tau_{xy} \\ s_y &= \sigma_y - \sigma_\mathrm{m} \qquad & s_{yz} &= \tau_{yz} \\ s_z &= \sigma_z - \sigma_\mathrm{m} \qquad & s_{zx} &= \tau_{zx} \end{aligned}$$

The material properties for the soil are tabulated in Table 12-1 below.:

TABLE 12-1: MATERIAL PROPERTIES FOR THE SOIL.

QUANTITY	NAME	EXPRESSION	UNIT
Young's modulus	E	10e6	Pa
Poisson's ratio	ν	0.3	-
Cohesion	c	10000	Pa
Friction angle	ϕ	35	deg
Specific weight	γ	18000	N/m^3

The model assumes that the tunnel behaves completely elastically and uses some typical material values for concrete:

TABLE 12-2: MATERIAL PROPERTIES FOR THE TUNNEL.

QUANTITY	NAME	EXPRESSION	UNIT
Young's modulus	E	25e9	Pa
Poisson's ratio	ν	0.33	-
Specific weight	γ	25000	N/m^3

CONSTRAINTS AND LOADS

- The lower horizontal boundary is restrained from moving in both the x and y directions, thereby simulating a rough and rigid underlying rock layer.
- The model includes only one half of the tunnel due to symmetry. Symmetry plane constraints are thus applied on the symmetry-cut boundaries.
- The left vertical boundary is assumed to be perfectly smooth and rigid. This is modeled by applying a constraint only in the horizontal direction, while allowing movement in the vertical direction, that is, a roller constraint (see also Figure 12-15).
- The specific weight of the soil is entered as a domain load in the negative y direction. The parametric solver ramps up the specific weight and the in-situ stress, which you enter an initial stress, during the elastic-perfectly plastic analysis in the first excavation stage of the analysis (solver parameter values `para` from 0 to 1). Originally, the soil carries the following in-situ stresses before the excavation:

$$\sigma_x = \lambda \cdot \sigma_y$$
$$\sigma_y = -\gamma \cdot y$$
$$\sigma_z = \lambda \cdot \sigma_y$$
$$\lambda = \frac{\nu}{(1-\nu)}$$

where γ denotes the specific weight of the soil, y is the vertical coordinate from the ground level, and ν is Poisson's ratio.

- The specific weight of the tunnel is ramped up and applied as a domain load in the negative y direction in the second stage of the model (solver parameter values `para` from 1 to 2).
- Additional pressure loads are also applied in the second stage. The first pressure load is ramped up to $5{\cdot}10^4$ Pa on the upper horizontal boundaries of the tunnel domain and a second pressure load is ramped up to $15{\cdot}10^4$ Pa on the inner lower horizontal boundaries.
- The vertical displacement of the lower horizontal edge of the tunnel, which is the common boundary with the soil domain, is set to be equal with the vertical displacement of the soil. There is however no horizontal displacement constraints on this boundary and thereby allowing a relative sliding between the tunnel and the soil.

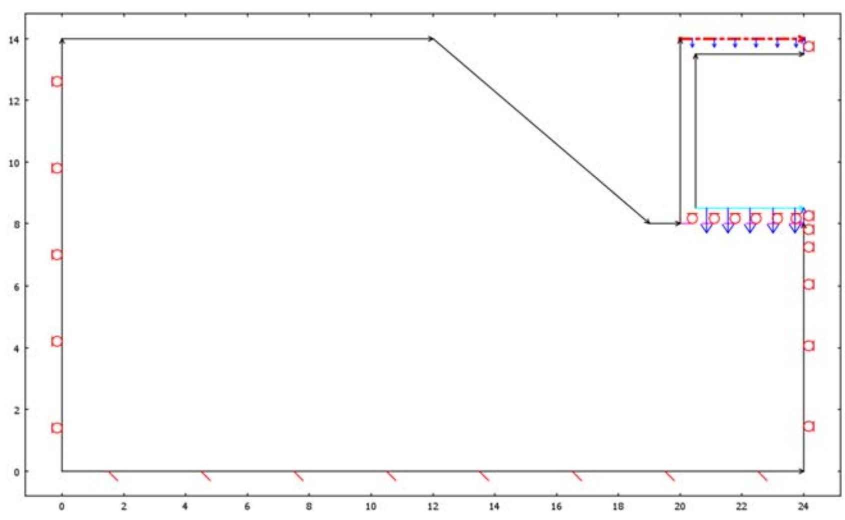


Figure 12-15: Boundary constraints and loads.

Modeling in COMSOL Multiphysics

The model includes two Plane Strain application modes. The first application mode is active in the soil domain and computes the elastic perfectly-plastic response of this domain. The response due to the excavation and the in-situ stress is calculated in the first stage of the model, where you ramp up the specific weight of the soil and the in-situ stresses using the parametric solver for parameter values between 0 and 1. This is a suitable method to achieve convergence for problems containing nonlinearities.The full specific weight and in-situ stresses are reached at parameter value 1 and kept constant at these levels for the following parameter steps.

The second stage computes the soil response due to the additional weight and stiffness from the tunnel and boundary loads on the tunnel. In this analysis the parametric solver ramps up the Young's modulus of the tunnel, the weight of the tunnel, and the boundary loads in the same way as in the first stage but with parameter values from 1 to 2. The ramping up of the Young's modulus from a nonsignificant value during the parameter values between 0 and 1 to full stiffness at parameter value 2 is due to convergence reasons. This is also the reason for ramping up the tunnel weight and boundary loads.

The second application mode is active in the tunnel domain and models the elastic deformations of the tunnel construction. The use of a separate application mode for the tunnel domain makes it possible to apply a tangential slip conditions between the soil domain and the tunnel domain.

The elastic perfectly-plastic Mohr-Coulomb material model is implemented with a user-defined yield function and a perfectly plastic hardening function. The yield function contains hydrostatic and equivalent deviatoric stresses as well as the material property constants α and K. You define both the stress variables and the material properties as scalar expression variables. A second set of scalar expressions of these variables use variables evaluated at the Gauss points. The second set of variables is for postprocessing purposes.

Convergence problems can occur in elasto-plastic problems like this. A couple of remedies to these problems are to use the mixed U-P formulation for nearly incompressible materials and to manually tune the parameter step-size settings. In this model you specify the initial parameter step and the minimum step size, while keeping the default setting for the maximum step size.

Results and Discussion

You can study the development of the stress levels, plastic regions, and deformations in the following figures. Figure 12-16 shows the stress levels and the plastic regions after the excavation. The red-colored areas indicate the plastic regions, and a contour plot shows the equivalent deviatoric stresses.

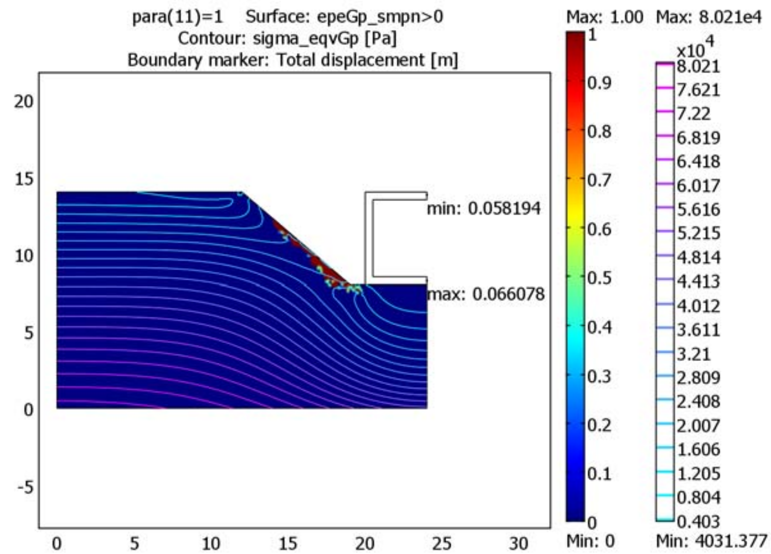


Figure 12-16: The equivalent deviatoric stresses and plastic regions due to the excavation.

Figure 12-17shows the equivalent deviatoric stresses in the soil domain as well as the maximum and minimum deformations when the surface pressures and the specific weight of the tunnel are applied. The stresses has reached the yield surface in the red colored areas.

The figure also shows that the maximum final vertical deformations of the tunnel construction are approximately 5 cm.

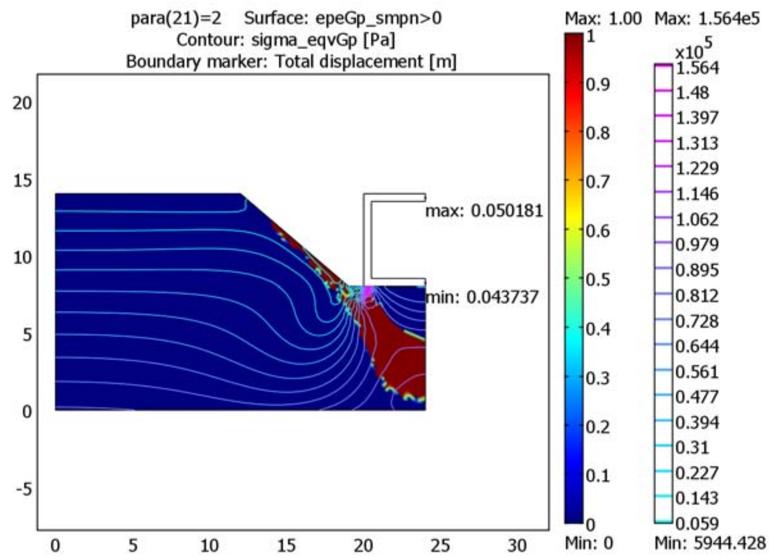


Figure 12-17: The equivalent deviatoric stresses and deformations at peak surface pressure.

References

1. I. Doltsinis., *Elements of Plasticity*, WIT Press, 2000.

2. O.C. Zienkiewicz, *The Finite Element Method*, McGraw-Hill, 1991.

3. W.F. Chen, *Nonlinear Analysis in Soil Mechanics*, Elsevier, 1990

Model Library path: `Structural_Mechanics_Module/Nonlinear_Material_Models/traffic_tunnel`

MODEL NAVIGATOR

1 Select **2D** from the **Space dimension** list on the **New** page in the **Model Navigator**.

2 Select **Structural Mechanics Module>Plane Strain>Static analysis elasto-plastic material**.

3 Click **OK**.

GEOMETRY MODELING

1 Add a rectangle by selecting from the **Draw** menu, **Specify Objects**, and then click **Rectangle**.

2 Type `24` in the **Width** edit field and type `14` in the **Height** edit field.

3 Click **OK**.

4 Next specify some lines. On the **Draw** menu point to **Specify Objects**, then click **Line**.

5 Enter the following coordinate pairs in the **x** and **y** edit fields for each line; click **OK** between each line.

LINE	X	Y
1	12 19	14 8
2	19 24	8 8

6 Select all geometry objects by pressing Ctrl+A.

7 Click the **Coerce to Solid** toolbar button.

8 Click the **Split Object** button.

9 Select the geometry object CO3 by clicking on it and delete it by pressing **Delete**.

10 Add a rectangle by selecting **Specify Objects** from the **Draw** menu and then clicking **Rectangle**.

11 In the **Width** edit field type `4` in the **Height** edit field type `6`.

12 In the **Position** area, in the **x** edit field type `20` and in the **y** edit field type `8`.

13 Click **OK**.

14 Add a second rectangle by selecting from the **Draw** menu, **Specify Objects**, and then click **Rectangle**.

15 Type `3.5` in the **Width** edit field and type `5` in the **Height** edit field.

16 In the **Position** area, type `20.5` in the **x** edit field and type `8.5` in the **y** edit field.

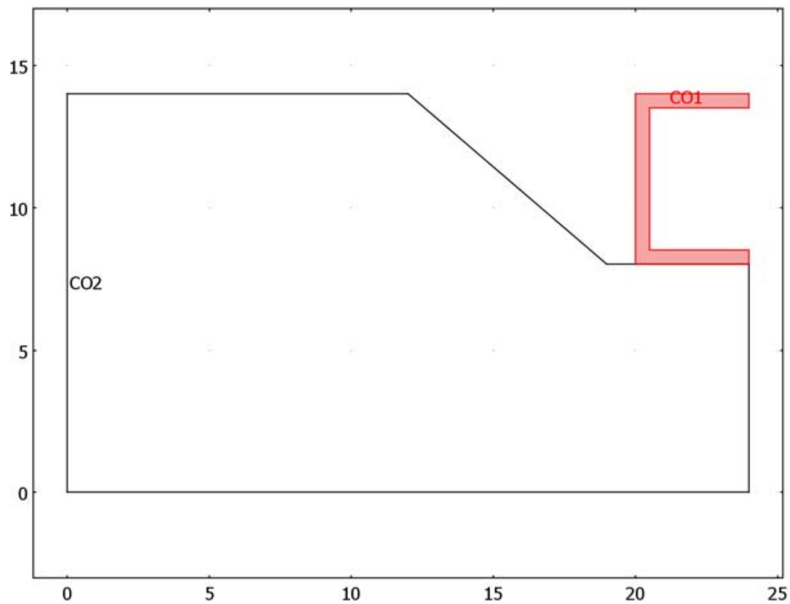
17 Click **OK**.

18 Click the **Create Composite Object** button on the Draw toolbar.

19 Write `R1-R2` in the **Set Formula** edit field.

20 Click **OK**.

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



OPTIONS AND SETTINGS

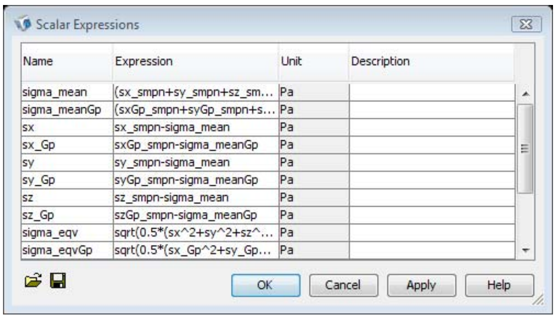
Scalar Expressions

1 Choose **Options>Expressions>Scalar Expressions**.

2 Specify expressions according to the following table:

NAME	EXPRESSION
`sigma_mean`	`(sx_smpn+sy_smpn+sz_smpn)/3`
`sigma_meanGp`	`(sxGp_smpn+syGp_smpn+szGp_smpn)/3`
`sx`	`sx_smpn-sigma_mean`
`sx_Gp`	`sxGp_smpn-sigma_meanGp`
`sy`	`sy_smpn-sigma_mean`
`sy_Gp`	`syGp_smpn-sigma_meanGp`
`sz`	`sz_smpn-sigma_mean`

NAME	EXPRESSION
sz_Gp	szGp_smpn-sigma_meanGp
sigma_eqv	sqrt(0.5*(sx^2+sy^2+sz^2)+sxy_smpn^2)
sigma_eqvGp	sqrt(0.5*(sx_Gp^2+sy_Gp^2+sz_Gp^2)+sxyGp_smpn^2)
alpha	tan(phi)/sqrt(9+12*(tan(phi))^2)
K	(3*c)/sqrt(9+12*(tan(phi))^2)
F	3*alpha*sigma_mean+sigma_eqv

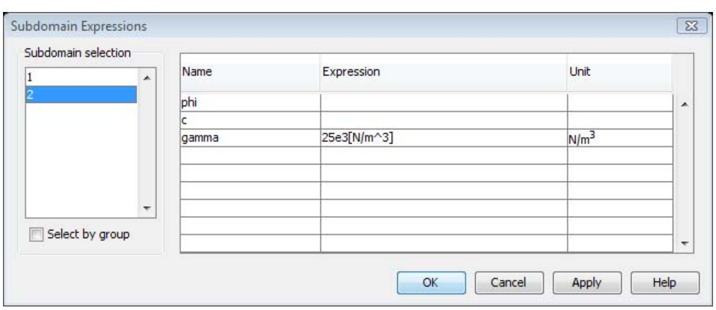


Subdomain Expressions

1 From the **Options** menu, choose **Expressions>Subdomain Expressions**.

2 Specify expressions according to the following table:

NAME	EXPRESSION IN SUBDOMAIN 1	EXPRESSION IN SUBDOMAIN 2
phi	35[deg]	
c	10000[Pa]	
gamma	18e3[N/m^3]	25e3[N/m^3]



PHYSICS SETTINGS

Boundary Settings

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

2 Specify boundary settings according to the following table:

	BOUNDARY 1	BOUNDARY 2	BOUNDARY 12
Page	Constraint	Constraint	Constraint
Constraint condition	Roller	Fixed	Symmetry plane

Subdomain Settings

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

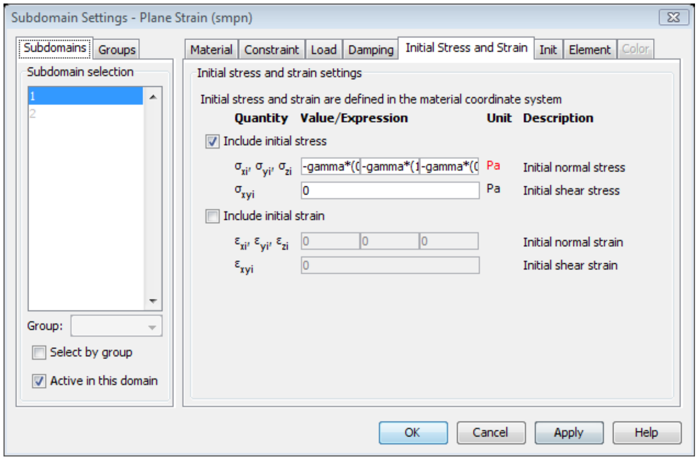
2 Select Subdomain 2 in the **Subdomain selection** list.

3 Clear the **Active in this domain** check box to deactivate this subdomain in this application mode.

4 Select Subdomain 1 in the **Subdomain selection** list.

5 Verify and enter subdomain data according to the following table:

	SUBDOMAIN 1	
Page	Material	
	Material model	Elasto-plastic material
	E	`10e6`
	ν	`0.3`
Page	Load	
	F_y	`-gamma*para*(para<=1)-gamma*(para>1)`
Page	Initial Stress and Strain	
	σ_{xi}	`-gamma*(0.3/(1-0.3))*(14-y)*para*(para<=1)-gamma*(0.3/(1-0.3))*(14-y)*(para>1)`
	σ_{yi}	`-gamma*(14-y)*para*(para<=1)-gamma*(14-y)*(para>1)`
	σ_{zi}	`-gamma*(0.3/(1-0.3))*(14-y)*para*(para<=1)-gamma*(0.3/(1-0.3))*(14-y)*(para>1)`
	σ_{xyi}	`0`



6 Select the **Use mixed U-P formulation (nearly incompressible material)** check box on the **Material** page.

7 Click the **Elasto-plastic material data** button.

8 Select **Perfectly plastic** from the **Hardening model** list in the **Elasto-plastic material settings** dialog box.

9 Select **User defined** from the **Yield function** list and type `F` in the **Yield function** edit field.

10 Type `K` in the **Yield stress level** edit field.

11 Click **OK** to close the **Elasto-plastic material settings** dialog box.

12 Click **OK** to close the **Subdomain Settings** dialog box.

MESH GENERATION

1 Click the **Initialize Mesh** toolbar button to generate the mesh.

2 Click the **Refine Mesh** toolbar button to refine the mesh.

MODEL NAVIGATOR

Next add a second Plane Strain application mode:

1 Select **Model Navigator** from the **Multiphysics** menu.

2 Select **2D** in the **Space dimension** list on the **New** page in the **Model Navigator**.

3 Select **Structural Mechanics Module>Plane Strain>Static analysis**.

4 Click **OK**.

PHYSICS SETTINGS

Subdomain Settings

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

2 Select Subdomain 1 in the **Subdomain selection** list.

3 Clear the **Active in this domain** check box to deactivate this subdomain in this application.

4 Enter subdomain data according to the following table.

	SUBDOMAIN 2	
Page	Material	
	Material model	Isotropic material
	E	`25e9*(para-1)*(para>1)+100`
	ν	`0.33`
Page	Load	
	F_y	`-gamma*(para-1)*(para>1)`

5 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Settings

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

2 Specify boundary settings according to the following tables:

	BOUNDARIES 13, 14
Page	Constraint
Constraint condition	Symmetry plane

	BOUNDARY 7		
Page	Constraint		
Constraint condition	Prescribed displacement	R_y	v

	BOUNDARY 8	
Page	Load	
	F_y	`-5e4*(para>1)*(para-1)`

	BOUNDARY 10	
Page	Load	
	F_y	`-15e4*(para>1)*(para-1)`

COMPUTING THE SOLUTION

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

2 Type the name `para` in the **Parameter name** edit field.

3 Type `0:0.1:2` in the **Parameter values** edit field.

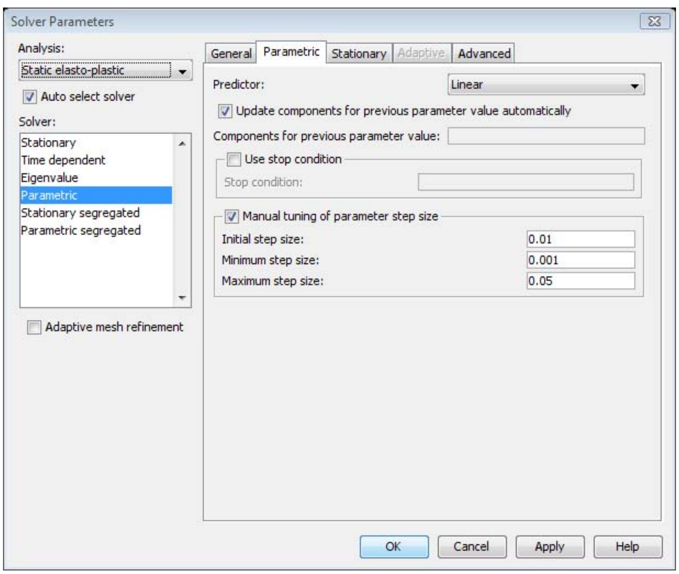
4 Click the **Parametric** tab.

5 Select the **Manual tuning of parameter step size** check box.

6 Type `0.01` in the **Initial step size** edit field.

7 Type `0.001` in the **Minimum step size** edit field.

8 Type `0.05` in the **Maximum step size** edit field.



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

10 Click the **Solve** toolbar button.

POSTPROCESSING AND VISUALIZATION

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

2 Select **1** from the **Parameter value** list on the **General** page.

3 Click the **Surface** tab.

4 Type `epeGp_smpn>0` in the **Expression** edit field to plot the areas where plastic deformation has occurred.

5 On the **Contour** page select the **Contour plot** check box.

6 In the **Expression** edit field type `sigma_eqvGp` to plot the equivalent deviatoric stress.

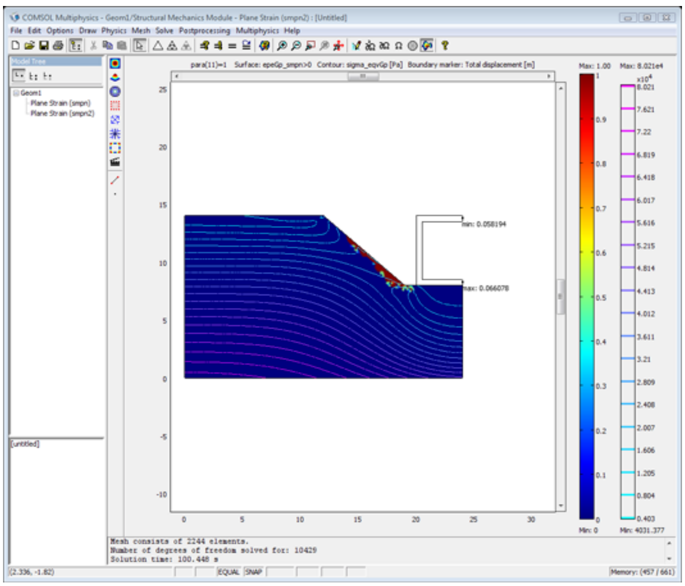
7 Click the **Max/Min** tab and select the **Max/Min marker** check box.

8 On the **Subdomain Data** page clear the **Subdomain max/min data** check box.

9 On the **Boundary Data** page select the **Boundary max/min data** check box.

10 From the **Predefined quantities** list select **Plane Strain (smpn2)>Total displacement**.

11 Click **OK**.



Flexible and Smooth Strip Footing on Stratum of Clay

Model Definition

A typical verification model for geotechnical problems is a shallow stratum layer of clay, depicted in Figure 12-18 below. A vertical load is applied on the clay stratum and the static response and the collapse load are studied.

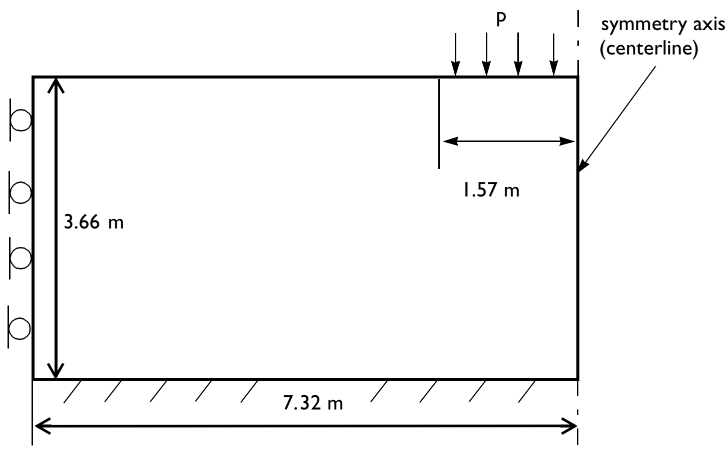


Figure 12-18: Dimensions, boundary conditions, and pressure load for the stratum of clay.

ANALYSIS TYPE

Yield Surface

Model the clay as a linear elastic-perfectly plastic material, and assume plane-strain conditions in this analysis.

The *yield surface, F*, for the Drucker-Prager material law is given by

$$F = \alpha \cdot I_1 + \sqrt{J_2} = K$$

where I_1 is the first stress invariant and J_2 is the second deviatoric stress invariant.

The first stress invariant is defined using the normal stress components:

$$I_1 = \sigma_x + \sigma_y + \sigma_z$$

The second stress invariant is defined by

$$I_2 = \sigma_x \cdot \sigma_y + \sigma_y \cdot \sigma_z + \sigma_z \cdot \sigma_x - \tau_{xy}^2$$

The second deviatoric stress invariant can be expressed using the first and the second stress invariants:

$$J_2 = \frac{1}{3} \cdot I_1^2 - I_2$$

If two-dimensional plane-strain conditions prevail the Drucker-Prager criterion becomes identical to the Mohr-Coulomb criterion if the material parameters α and K are given by (see Ref. 1)

$$\alpha = \frac{\tan\phi}{\sqrt{(9 + 12 \cdot \tan^2\phi)}}$$

$$K = \frac{3 \cdot c}{\sqrt{(9 + 12 \cdot \tan^2\phi)}}$$

The model employs the Mohr-Coulumb yield condition with an associated flow rule and a nonassociated flow rule.

Flow Rule

The flow rule defines the relation between the plastic strain increment in a given direction and the current level of stress in the same direction. The relation reads

$$d\varepsilon_{ij} = d\lambda \frac{\partial Q}{\partial \sigma_{ij}}$$

where $d\lambda$ is the plastic multiplier and Q is the plastic potential. If the yield surface, F, and the plastic potential, Q, are identical, that is, if $F = Q$, then the rule is called an *associated* flow rule, otherwise a *nonassociated* flow rule.

The associated flow does unfortunately not always describe the real physical process in a sufficiently realistic manner, especially if the material is pressure sensitive like for example, soil. Usually, the associated flow rule overestimates the volumetric plastic strains.

The nonassociated flow rule case uses a von Mises type of function for the plastic potential in this model. This function does not include volumetric plastic strains. The plastic potential can thus be written as

$$Q = \sqrt{J_2}$$

MATERIAL PROPERTIES

- Young's modulus, $E = 207$ MPa, and Poisson's ratio $\nu = 0.3$.
- Cohesion $c = 69$ MPa and angle of internal friction, $\phi = 20$ degrees.

CONSTRAINTS AND LOADS

- The clay layer is supported by a rigid and perfectly rough base. Therefore, fix the lower horizontal boundary.
- The left vertical boundary is perfectly smooth and a roller boundary condition is suitable to use at this boundary.
- Model only the left half of the domain due to symmetry reasons. Thus, use symmetry boundary condition at the right vertical boundary.
- The stratum is subjected to a footing that you can consider to be flexible and smooth. The width of the strip footing is 3.14 m. Gradually increase the footing pressure until the clay layer reaches the collapse load.

This example is derived from *Nonlinear Analysis in Soil Mechanics* by W.F. Chen and E. Mizuno (Ref. 1).

Results and Discussion

You can study in Figure 12-19 the load-displacement curves for both the associated case and the nonassociated case. The figure shows the applied footing pressure versus the centerline displacement (directly beneath the footing's center) in the y direction. The dashed line shows the load-displacement curve for the nonassociated flow rule case and the solid line shows the load-displacement curve for associated flow rule case.

The model uses the SI unit system. The load-displacement curves in Ref. 1 are however given in psi (pounds per square inch) and inches. To facilitate comparisons, the load-displacement plots use these units.

The curves are identical up to 40 psi because the whole domain is still within the elastic region. When the pressure increases the behavior diverges. The curve for the associated flow rule bends at approximately 150 psi and reaches the collapse load at approximately

160 psi. The curve for the nonassociated case deviates gradually from the associated case and reaches a collapse load at approximately 146 psi. The collapse load deviates only by approximately 3% from the solution given in Ref. 1.

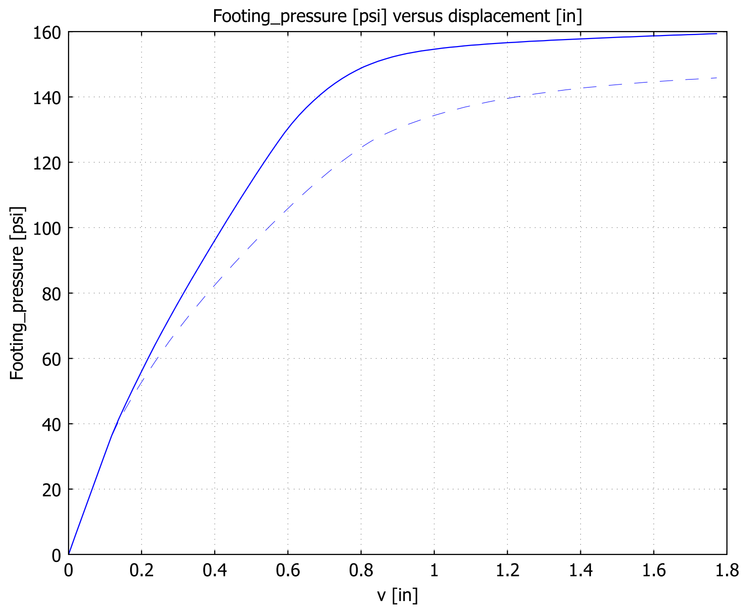


Figure 12-19: Footing pressure vs. vertical displacement for the nonassociated case (dashed line) and the associated case (solid line).

Modeling in COMSOL Multiphysics

Implement the Mohr-Coulomb elastic-perfectly plastic material model with a user-defined yield function and a perfectly plastic hardening function. The yield function contains the first stress invariant, I_1, and the second deviatoric stress invariant, J_2, as well as the material property constants α and K. You define the stress variables and the material properties as scalar expression variables and constants.

Model the associated flow rule case in the graphical user interface. For the nonassociated flow rule case, on the other hand, you need to model on the command line. The workflow for the nonassociated flow rule case:

1 Export the FEM structure for the associated case from the graphical user interface to the command line.

2 Edit the plastic potential definition.

3 Solve and postprocess the model on the command line.

A suitable modeling technique in a case like this, where the relation between the applied load and the displacement is highly nonlinear, is to use an algebraic equation that controls the applied pressure so that the model reaches the desired displacement increments. The ODE functionality in COMSOL Multiphysics provides the means for entering the algebraic equation, and the parametric solver steps up the desired vertical displacement.

Convergence problems can occur in elasto-plastic problems like this. If this happens, try scaling the solution components manually.

Reference

1. W.F. Chen and E. Mizuno, *Nonlinear Analysis in Soil Mechanics*, Elsevier, 1990.

Model Library path: `Structural_Mechanics_Module/`
`Nonlinear_Material_Models/flexible_footing`

Modeling Using the Graphical User Interface

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

2 Select **Structural Mechanics Module>Plane Strain>Static analysis elasto-plastic material**.

3 From the **Element** list select **Lagrange - Linear**.

4 Click **OK**.

Options and Settings

1 From the **Options** menu select **Constants**. Enter constant names, expressions, units and descriptions (optional) according to the following table:

NAME	EXPRESSION	DESCRIPTION
E	2.07e8[Pa]	Young's modulus
nu	0.3	Poisson's ratio
phi	20/180*pi	Angle of internal friction
c	69e3[Pa]	Cohesion

NAME	EXPRESSION	DESCRIPTION
alpha	tan(phi)/sqrt(9+12*(tan(phi))^2)	Drucker-Prager material constant
K	(3*c)/sqrt(9+12*(tan(phi))^2)	Drucker-Prager material constant

2 Click **OK**.

3 From the **Options** menu select **Expressions>Scalar Expressions**. Enter expression names, expressions, and descriptions (optional) according to the following table:

NAME	EXPRESSION	DESCRIPTION
I1	sx_smpn+sy_smpn+sz_smpn	First stress invariant
I2	sx_smpn*sy_smpn+ sy_smpn*sz_smpn+ sz_smpn*sx_smpn-sxy_smpn^2	Second stress invariant
J2	1/3*I1^2-I2	Second deviatoric stress invariant
F	(alpha*I1+sqrt(J2))	Yield surface
Footing_pressure	-Pressure/1[psi]	Pressure in psi

Note that the unit for the variable `Footing_pressure` has red colored brackets because an ODE variable (`Pressure)` is used in the expression for this variable. Dividing the `Pressure` variable with `1[psi]` converts the value for this variable to psi.

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

GEOMETRY MODELING

1 Shift-click the **Rectangle/Square** button on the Draw toolbar. In the **Width** edit field type `7.32` and in the **Height** edit field type `3.66` Click **OK**.

2 Shift-click the **Point** button. In the **x** edit field type `5.75` and in the **y** edit field type `3.66`, then click **OK**.

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

PHYSICS SETTINGS

Boundary Settings

1 From the **Physics** menu select **Boundary Settings**.

2 Specify boundary settings according to the following table:

BOUNDARY	PAGE	CONSTRAINT CONDITION	QUANTITY	VALUE
1	Constraint	Roller		
2	Constraint	Fixed		
5	Constraint	Symmetry plane		
4	Load		`Fy`	`Pressure`

Subdomain Settings

1 From the **Physics** menu select **Subdomain Settings**.

2 Select Subdomain 1, then verify and specify data on the **Material** page according to the following table:

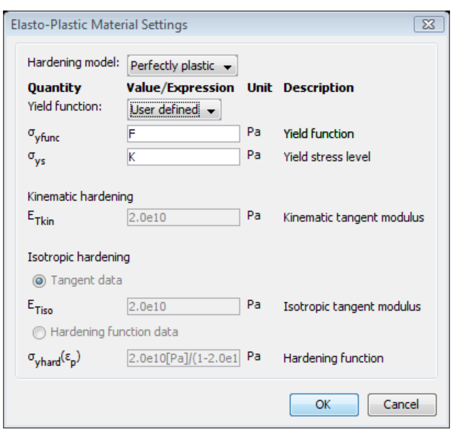
SETTINGS	VALUE
Material model	Elasto-plastic material
E	E
ν	nu

3 Click the **Elasto-plastic material data** button to open the **Elasto-plastic material settings** dialog box.

4 From the **Hardening model** list select **Perfectly plastic**.

5 From the **Yield function** list select **User defined**.

6 In the σ_{yfunc} edit field type `F` and in the σ_{ys} edit field type `K`.



7 Click **OK**.

8 Click **OK** to close the **Subdomain Settings** dialog box.

Integration Coupling Variable

1 From the **Options** menu select **Integration Coupling Variables>Point Variables**.

2 Select Point 5.

3 Type `vc` in the **Name** edit field and `v` in the **Expression** edit field.

4 Click **OK** to close the **Point Integration Variables** dialog box.

By defining the above point integration coupling variable *vc* the deformation in *y* direction at Point 5 is made available for the next step where an algebraic equation is created.

Global Equations

1 Select **Global Equations** from the **Physics** menu.

2 Enter `Pressure` in the **Name** edit field and `vc-para` in the **Equation** edit field.

3 Click **OK**.

MESH GENERATION

1 From the **Mesh** menu, choose **Free Mesh Parameters**.

2 Select **Fine** from the **Predefined mesh sizes** list.

3 Click the **Remesh** button and then click **OK**.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 In the **Parameter name** edit field type the name `para`.

3 In the **Parameter values** edit field type `0:-0.0005:-0.045`. Click **OK**.

4 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

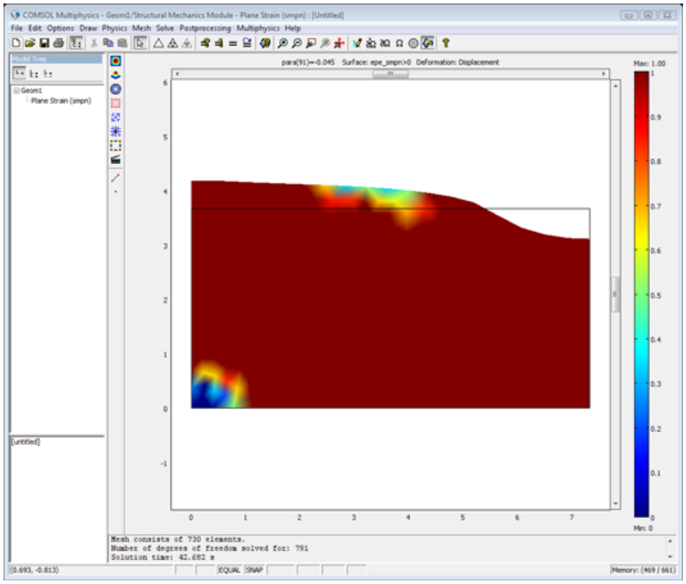
1 Click the **Plot Parameters** button on the Main toolbar.

2 Select the **Surface** and **Deformed shape** check boxes in the **Plot type** area on the **General** page.

3 Click the **Surface** tab.

4 In the **Expression** edit field type `epe_smpn>0`.

5 Click **OK**.



6 From the **Postprocessing** menu select **Domain Plot Parameters**.

7 Click the **Point** tab.

8 Select Point 5.

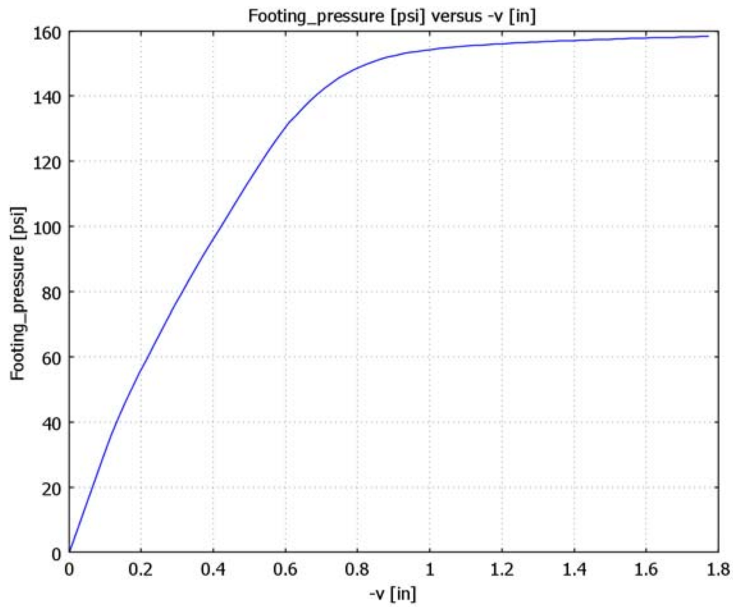
9 Into the **Expression** edit field enter `Footing_pressure`.

10 In the **x-axis data** frame select **Expression**.

11 Click the **Expression** button and enter `-v` in the **Expression** edit field.

12 From the **Unit** list select **in**, then click **OK**.

13 Click **OK**.



While the graphical user interface supports the associated flow rule, you need to define, solve, and postprocess the nonassociated flow rule model on the command line. You need COMSOL Script or MATLAB for this part of the simulation. The following lines change the plastic potential to a von Mises function, solve the problem, and plot the footing pressure as function of the displacement in the same figure as for the associated case (make sure that the figure window for the previous results plot is open):

1 Select **Export** and **FEM Structure as 'fem'** from the **File** menu.

2 Enter the following lines at the command line prompt or run the file `flexible_footing_nonassoc.m`. The M-file is located in the subfolder denoted `sme` in your COMSOL installation directory.

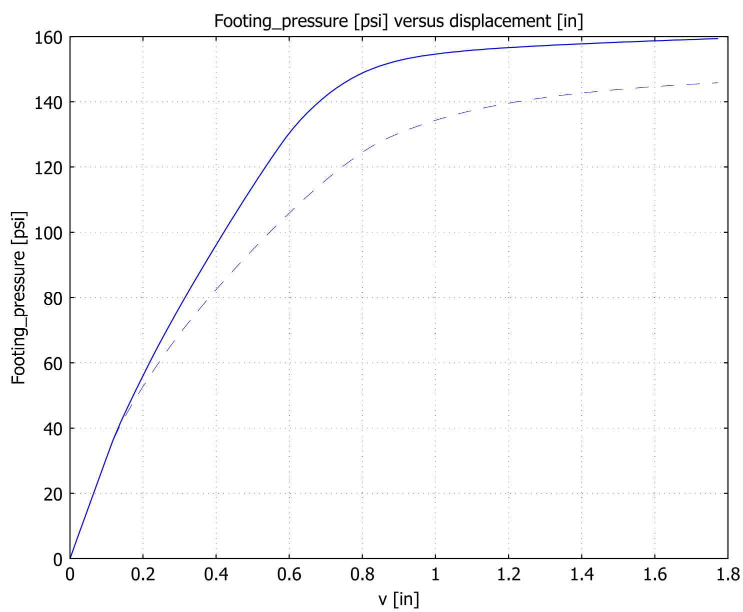
```
G = fem.elemmph(3);
G{1}.geomdim{1}{3}.G{1}{1} = 'diff(mises_smpn,sx_smpn)';
G{1}.geomdim{1}{3}.G{1}{2} = 'diff(mises_smpn,sy_smpn)';
G{1}.geomdim{1}{3}.G{1}{3} = 'diff(mises_smpn,sz_smpn)';
G{1}.geomdim{1}{3}.G{1}{4} = '0.5*diff(mises_smpn,sxy_smpn)';
fem.elemmph(3)=G;

% Extend mesh
fem.xmesh=meshextend(fem);
```

```
% Solve problem
fem.sol=femnlin(fem, ...
'u',0, ...
'solcomp',{'u','v','Pressure'}, ...
'outcomp',{'epx_smpn','u','epe_smpn','epz_smpn','v',...
'epxy_smpn','epy_smpn','Pressure'}, ...
'pname','para', ...
'pinitstep',1e-4, ...
'pminstep',1e-6, ...
'pmaxstep',0.01,...
'plist',[0:-0.0005:-0.045], ...
'uscale',{'u','0.01','v','0.01'}, ...
'hnlin','on');

% Plot in cross-section or along domain
hold on
postcrossplot(fem,0,[5], ...
'pointdata','Footing_pressure', ...
'pointxdata',{'-v','unit','in'}, ...
'title','Footing_pressure [psi] versus displacement [in]', ...
'axislabel',{'v [in]','Footing_pressure [psi]'}, ...
'refine','auto','linstyle','--');
```

The following figure shows the finished plot.



13

Piezoelectricity Models

The piezoelectric effect can be seen as a transfer of electric to mechanical energy and vice-versa. It is observed in many crystalline materials. Some materials, such as quartz, Rochelle salt, and lead titanate zirconate ceramics, display the phenomenon strongly enough to use it.

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 electric field induces a deformation of the crystal.

This chapter shows the application of the Structural Mechanics Module for modeling the piezoelectric effect using a static linear analysis. A 3D example describes modeling of the inverse piezoelectric effect.

An introductory piezoelectric model of a piecoceramic tube is also available. You find the documentation for this model in the *Structural Mechanics Module User's Guide*.

Piezoceramic Tube

Introduction

This example performs a static 2D axisymmetric analysis of a piezoelectric actuator using the 2D Piezo Axial Symmetry application mode. It models a radially polarized piezoelectric tube as described by S. Peelamedu and others (Ref. 1). One application area where radially polarized tubes are employed is in nozzles for fluid control in inkjet printers.

Model Definition

GEOMETRY

The tube has a height of 0.62 mm and an inner and outer radius of 0.38 mm and 0.62 mm, respectively.

MATERIAL

The material properties for the piezoceramic (PZT-5H) are:

$$c = \begin{bmatrix} 127 & 80.2 & 84.7 & 0 & 0 & 0 \\ & 127 & 84.7 & 0 & 0 & 0 \\ & & 117 & 0 & 0 & 0 \\ & & & 23.0 & 0 & 0 \\ & \text{sym} & & & 23.0 & 0 \\ & & & & & 23.5 \end{bmatrix} \text{GPa}$$

$$e = \begin{bmatrix} 0 & 0 & 0 & 0 & 17.03448 & 0 \\ 0 & 0 & 0 & 17.03448 & 0 & 0 \\ -6.22812 & -6.22812 & -23.2403 & 0 & 0 & 0 \end{bmatrix} \text{C/m}^2$$

$$\varepsilon = \begin{bmatrix} 1.5 & 0 & 0 \\ 0 & 1.5 & 0 \\ 0 & 0 & 1.3 \end{bmatrix} \cdot 10^{-8} \ \text{F/m}$$

BOUNDARY CONDITIONS

This model studies two cases, each of which studies different boundary conditions. Case 1 represents the inverse piezoelectric effect, and case 2 represents the direct piezoelectric effect.

Case 1:

- Structural mechanics boundary condition—constrain the bottom surface from moving axially (in the z-direction).
- Electrostatics boundary condition—apply a 1 V potential difference between the tube's inner and outer surfaces.

Case 2:

- Structural mechanics boundary condition—constrain the bottom surface from moving axially (in the z direction), but also add an internal fluid pressure of 0.1 MPa.
- Electrostatics boundary condition—ground the inner and outer surfaces.

Results and Discussion

The image in Figure 13-1 shows the deformation due to the applied voltage difference in Case 1. For the same case, Figure 13-2 shows the radial displacement as a function of the tube thickness at the top boundary.

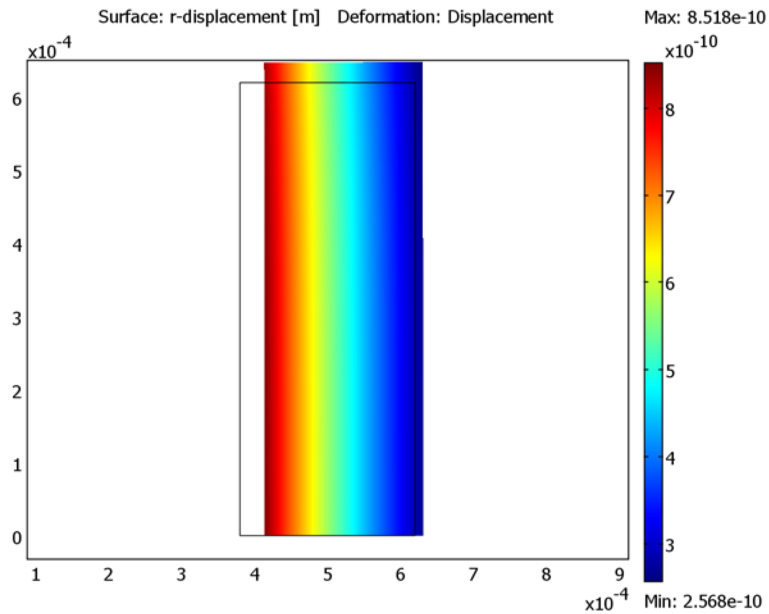


Figure 13-1: The deformed shape and radial displacement of a piezoceramic-tube actuator due to the radial electric field for Case 1.

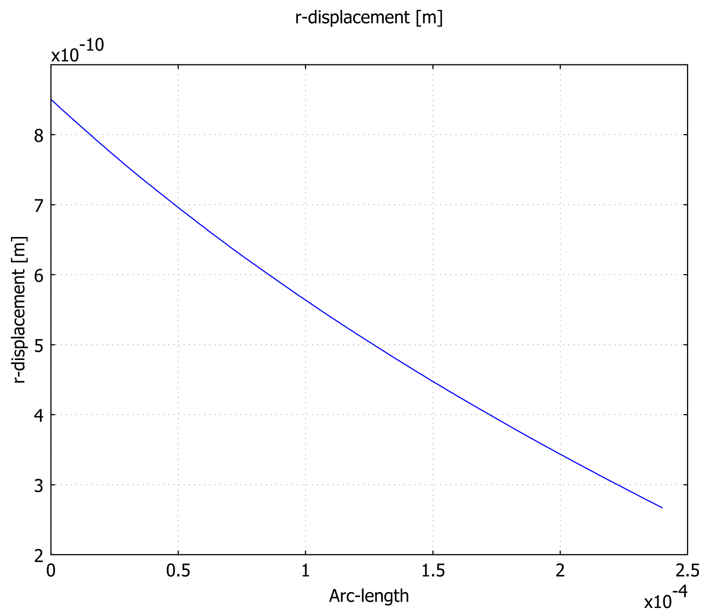


Figure 13-2: The radial displacement as a function of the tube thickness due to the radial electric field (Case 1).

For the second case, Figure 13-3 shows the deformed shape in Case 2. For that same case, Figure 13-4 shows the radial displacement as a function of tube thickness. These results show good agreement with those from S. Peelamedu (Ref. 1).

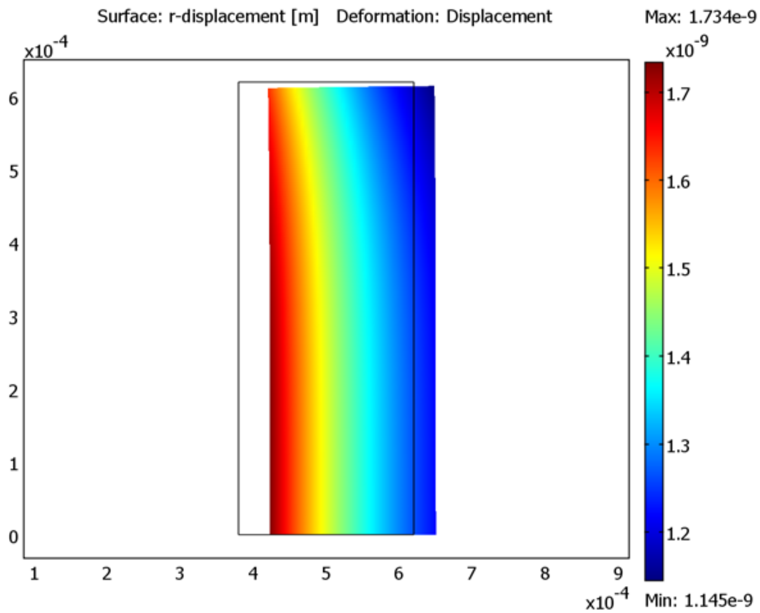


Figure 13-3: The deformed shape and radial displacement in a piezoceramic-tube actuatordue to an internal pressure of 0.1 MPa (Case 2).

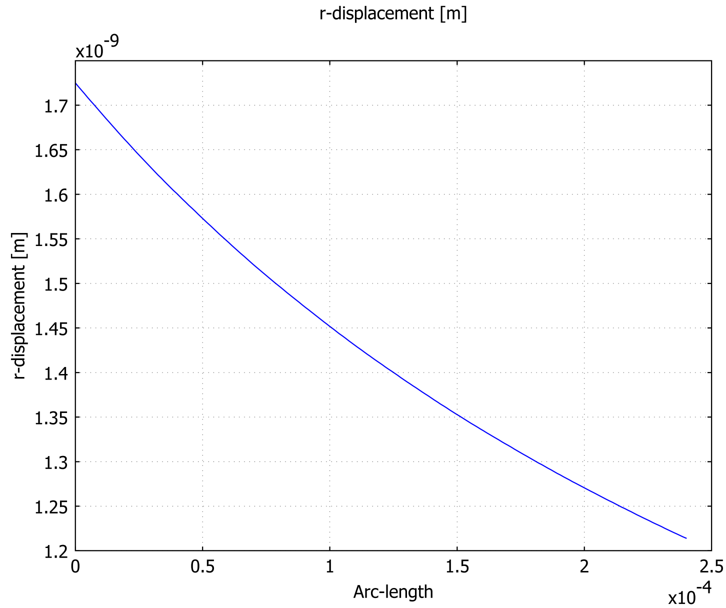


Figure 13-4: The radial displacement as function of the tube thickness due to an internal pressure of 0.1 MPa (Case 2).

Modeling in COMSOL Multiphysics

CONSTITUTIVE EQUATION AND MATERIAL DATA

You specify the material parameters for the piezoelectric material in the **Subdomain Settings** dialog box in the piezoelectric application mode. Then select the stress-charge form for the constitutive equation because this suits the form in which you give the material data. Further, enter the elasticity matrix into the c_e matrix, enter the piezoelectric coupling matrix into the e matrix, and enter the relative permittivities into the ε_{rS} matrix.

You enter the material properties in this example such that the polarization is in the z direction (in a 3D Cartesian coordinate system), which is a common orientation for published material data. This orientation means that you must rotate the material so that its polarization direction is aligned with the r-direction (radially polarized). To do so, use the material-orientation feature in the Piezo Axial Symmetry application mode. By selecting the material orientation as the zx-plane, you rotate the material so that its

z direction is aligned with the r direction of the model, and the material's x direction is aligned with the model's z direction.

The piezoceramic material in this example (PZT-5H) is a transversely isotropic material, which is a special class of orthotropic materials. Such a material has the same properties in one plane (isotropic behavior) and different properties in the direction normal to this plane. Thus you can use either the zx-plane material orientation or the zy-plane material orientation; both give the same solution.

MESHING

The rectangular geometry is well suited for quadrilaterals, and the model uses a 6-by-6-element grid.

Reference

1. S. Peelamedu et al., *Numerical Approach for Axisymmetric Piezoceramic Geometries towards Fluid Control Applications*, Univ. Toledo, OH, Mechanical, Industrial and Manufacturing Engineering Dept., 2000.

Model Library path: `Structural_Mechanics_Module/Piezoelectric_Effects/piezoceramic_tube`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Start COMSOL Multiphysics. This invokes the **Model Navigator**, which you can also open from an already running COMSOL Multiphysics session by choosing **New** from the **File** menu.
2. Select **Axial Symmetry (2D)** from the **Space dimension** list.
3. Select **Structural Mechanics Module>Piezoelectric Effects>Piezo Axial Symmetry>Static analysis** from the list of application modes.
4. Click **OK** to close the **Model Navigator**. Note that this gives you second-order elements, **Lagrange - Quadratic**, by default.

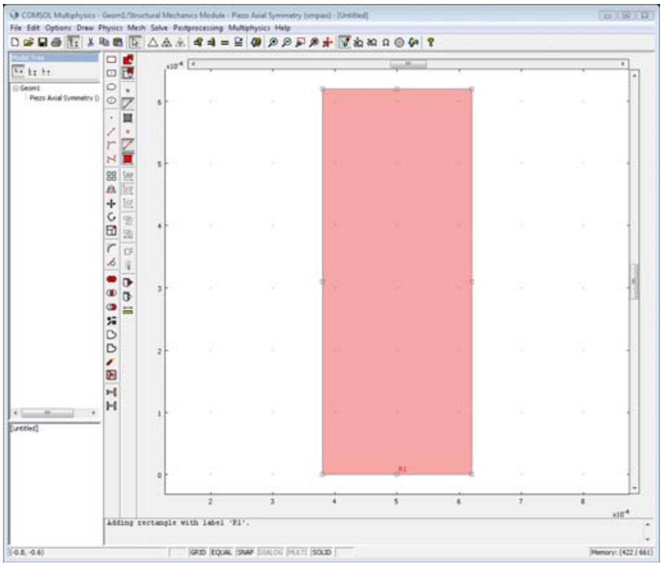
GEOMETRY MODELING

Define the model geometry in Draw mode.

1 Shift-click the **Rectangle/Square** button in the Draw toolbar to open the **Rectangle** dialog box.

2 Type `0.24e-3` in the **Width** edit field and `0.62e-3` in the **Height** edit field. In the **Position** area, type `0.38e-3` in the **r** edit field and `0` in the **z** edit field. This creates a rectangle with opposite corners at (0.38e-3, 0) and (0.62e-3, 0.62e-3).

3 Click the **Zoom Extents** button.



PHYSICS SETTINGS FOR MODEL CASE 1

Boundary Conditions

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

2 Select Boundary 2 from the **Boundary selection** list.

3 Select **Roller** from the **Constraint condition** list on the **Constraint** page.

4 Select all boundaries by pressing Ctrl+A.

5 Select **Zero charge/Symmetry** from the **Boundary condition** list on the **Electric BC** page.

6 Select Boundary 1 and then select **Ground** from the **Boundary condition** list.

7 Select Boundary 4 and then select **Electric potential** from the **Boundary condition** list and type `1` in the **Electric potential** edit field.

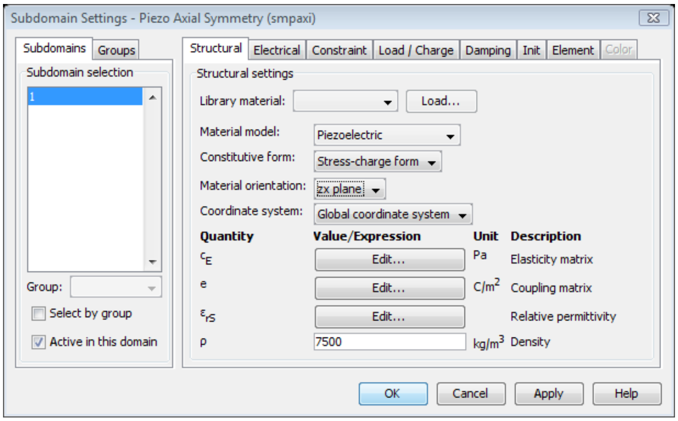
8 Click **OK** to close the dialog box.

Subdomain Settings

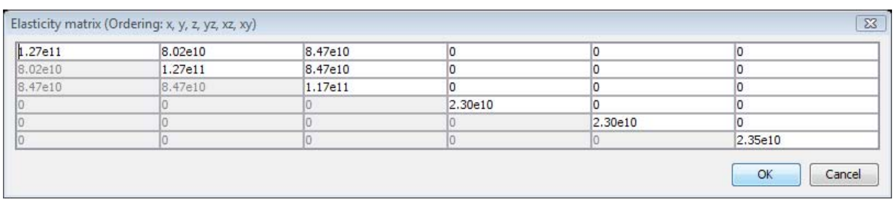
1 Open the **Subdomain Settings** dialog box by selecting **Subdomain Settings** from the **Physics** menu.

2 Select Subdomain 1 and click the **Structural** tab.

3 Select **zx plane** from the **Material orientation** list.



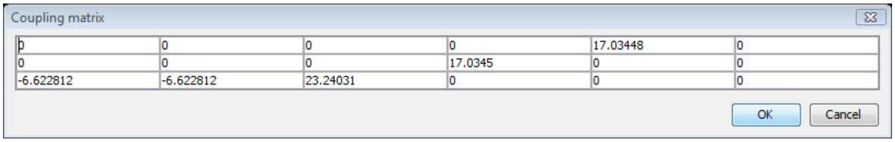
4 Click the **Edit** button associated with c_E and enter the following values into the **Elasticity matrix** dialog box; when done, click **OK**.



1.27e11	8.02e10	8.47e10	0	0	0
	1.27e11	8.47e10	0	0	0
		1.27e11	0	0	0
			2.30e10	0	0
				2.30e10	0
					2.30e10

Entries for the elasticity matrix.

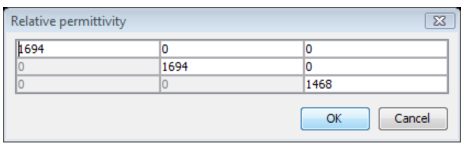
5 Click the **Edit** button associated with **e** and enter the following values into the **Coupling matrix** dialog box; when done, click **OK**.



0	0	0	0	17.03448	0
0	0	0	17.03448	0	0
-6.22812	-6.22812	-23.2403	0	0	0

Entries for the coupling matrix.

6 Click the **Edit** button associated with ε_{rS} and enter the following values into the **Relative permittivity matrix** dialog box; when done, click **OK**.



1694		
	1694	
		1468

Entries for the relative permittivity matrix.

7 Click **OK** again to close the **Subdomain Settings** dialog box.

MESH GENERATION

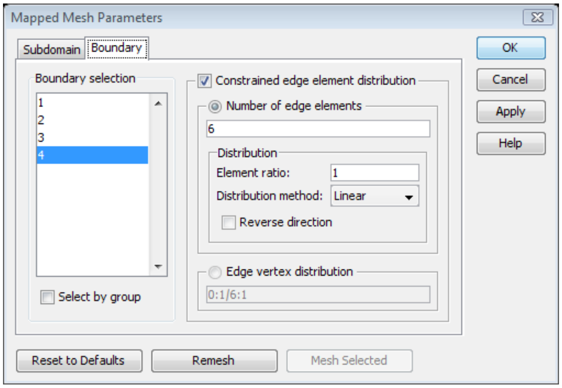
1 From the **Mesh** menu choose **Mapped Mesh Parameters**.

2 Click the **Boundary** tab.

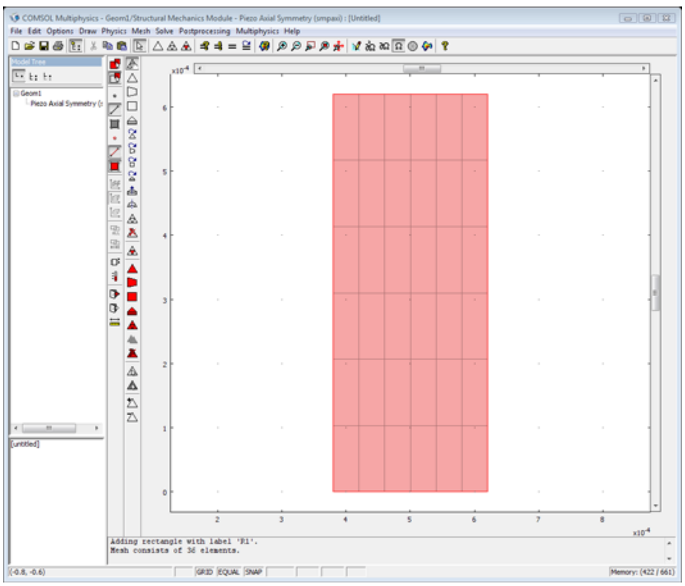
3 Select Boundary 1 and click the **Constrained edge element distribution** button.

4 Enter 6 in the **Number of edge elements** edit field.

5 Repeat this for the remaining three boundaries.



6 Click the **Remesh** button.



The meshed geometry for the piezoceramic-tube actuator.

7 Click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 Choose **Plot Parameters** from the **Postprocessing** menu to open the **Plot Parameters** dialog box.
2 On the **General** page, find the **Plot type** area, then select the **Deformed shape** and **Surface** check boxes.
3 Click the **Surface** tab.
4 Select **r-displacement** from the **Predefined quantities** list and click **OK** (this step reproduces Figure 13-1).
5 Choose **Cross-Section Plot Parameters** from the **Postprocessing** menu.
6 Click the **Line/Extrusion** tab.
7 Select **r-displacement** from the **Predefined quantities** list.
8 Go to the **Cross-section line data** area. In the **r0** edit field enter `3.8e-4`. Similarly set **r1** to `6.2e-4` and then both **z0** and **z1** to `3e-4`.
9 Click **OK** (this step reproduces Figure 13-2).

PHYSICS SETTINGS FOR MODEL CASE 2

Boundary Conditions

1 Open the **Boundary Settings** dialog box by selecting **Boundary Settings** from the **Physics** menu. In this dialog box you can select boundaries and enter expressions for boundary conditions.
2 Select Boundary 1 from the **Boundary selection** list.
3 Enter `0.1e6` in the $\mathbf{F_r}$ edit field on the **Load** page.
4 Select Boundaries 1 and 4 and then select **Ground** from the **Boundary condition** list on the **Electric BC** page.
5 Select Boundaries 2 and 3 and then select **Zero charge/Symmetry** from the **Boundary condition** list on the **Electric BC** page.
6 Click **OK** to close the dialog box.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 Choose **Plot Parameters** from the **Postprocessing** menu to open the **Plot Parameters** dialog box.

2 On the **General** page select the **Deformed shape** and **Geometry edges** check boxes in the **Plot type** area.

3 Click the **Surface** tab and select **r-displacement** from the **Predefined quantities** list and click **OK** (this reproduces Figure 13-3).

4 Choose **Cross-Section Plot Parameters** from the **Postprocessing** menu.

5 Click the **Line/Extrusion** tab.

6 Select **r-displacement** from the **Predefined quantities** list.

7 Go to the **Cross-section line data** area. In the **r0** edit field enter `3.8e-4`. Similarly set **r1** to `6.2e-4` and then both **z0** and **z1** to `3e-4`.

8 Click **OK** (this reproduces Figure 13-4).

Piezoelectric Shear Actuated Beam

Introduction

This example performs a static analysis on a piezoelectric actuator based on the movement of a cantilever beam, using the static 3D Piezo Solid application mode. Inspired by work done by V. Piefort (Ref. 1) and A. Benjeddou (Ref. 2), it models a sandwich beam using the shear mode of the piezoelectric material to deflect the tip.

Model Definition

GEOMETRY

The model consists of a sandwiched cantilever beam 100 mm long; it consists of a rigid foam core 2 mm thick sandwiched by two 8-mm thick aluminum layers. Further, the device replaces part of the foam core with a 10-mm long piezoceramic actuator that is positioned between z = 55 mm and z = 65 mm (Figure 13-5). The cantilever beam is orientated along the x-axis.

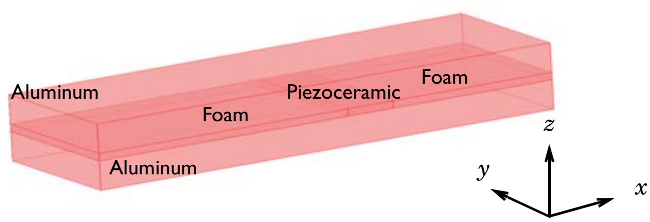


Figure 13-5: In this geometry for the shear bender note that a piezoceramic material replaces part of the foam core.

BOUNDARY CONDITIONS

- The structural mechanics boundary conditions for this model are that the cantilever beam is fixed at its surfaces at x = 0 and that all other surfaces are free.
- The electrostatic boundary conditions for this model are that the system applies a 20 V potential difference between the top and bottom surfaces of the piezoceramic subdomain. This gives rise to an electric field perpendicular to the poling direction (x-direction) and thus induces a transverse shear strain.

MATERIAL PROPERTIES

The following table lists the material properties for the aluminum layers and the foam core:

PROPERTY	ALUMINUM	FOAM
E	70 GPa	35.3 MPa
ν	0.345	0.383
ρ	2690 kg/m^3	32 kg/m^3

The matrices later in the section contain the material properties for the piezoceramic (PZT-5H): c_E is the elasticity matrix, e is the piezoelectric coupling matrix, and ε is the absolute permittivity matrix.

Note that it is necessary to recalculate the absolute permittivity matrix using the permittivity of vacuum ε_0 to the relative permittivity matrix.

Note also that the order of the material properties is such that the polarization direction is in the z direction. The polarization direction of the piezoceramic material in this model is however aligned with the x-axis and thus a local coordinate system must be used in the material settings to rotate the piezoceramic material.

$$c_E = \begin{bmatrix} 126 & 79.5 & 84.1 & 0 & 0 & 0 \\ & 126 & 84.1 & 0 & 0 & 0 \\ & & 117 & 0 & 0 & 0 \\ & & & 23.0 & 0 & 0 \\ & \text{sym} & & & 23.0 & 0 \\ & & & & & 23.3 \end{bmatrix} \text{GPa}$$

$$e = \begin{bmatrix} 0 & 0 & 0 & 0 & 17 & 0 \\ 0 & 0 & 0 & 17 & 0 & 0 \\ -6.5 & -6.5 & 23.3 & 0 & 0 & 0 \end{bmatrix} \text{C/m}^2$$

$$\varepsilon = \begin{bmatrix} 1.503 & 0 & 0 \\ 0 & 1.503 & 0 \\ 0 & 0 & 1.3 \end{bmatrix} \cdot 10^{-8} \text{ F/m}$$

Results

The shear deformation of the piezoceramic core layer and the flexible foam layer induce the bending action. Figure 13-6 shows the resulting tip deflection. The model calculates this deflection as 83 nm, which is in good agreement with the work in Ref. 1 and Ref. 2.

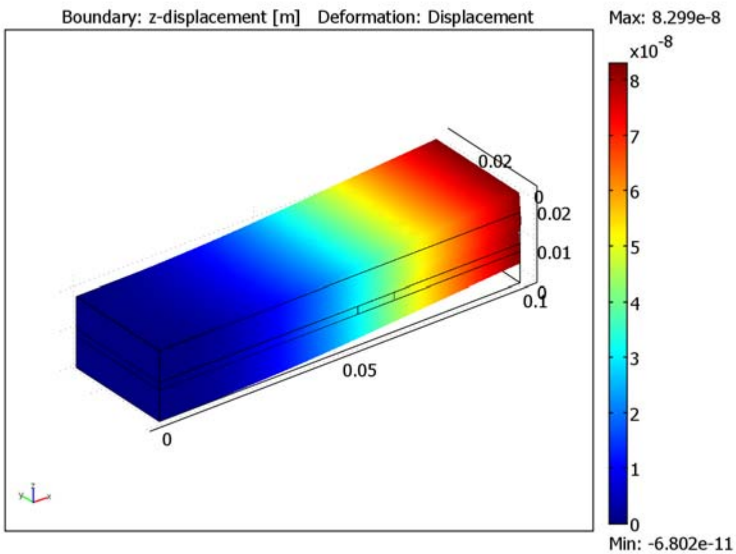


Figure 13-6: Tip deflection with the piezoceramic positioned at z = 60 mm.

Modeling in COMSOL Multiphysics

CONSTITUTIVE EQUATION AND MATERIAL DATA

You specify the material parameters for the piezoelectric material in the **Subdomain Settings** dialog box in the corresponding piezoelectric application mode. You select the stress-charge form for the constitutive equation because it suits the form in which you give the material data. Enter data for the elasticity matrix into the c_e matrix; enter data for the piezoelectric coupling matrix into the e matrix; and enter the relative permittivities into the ε_{rs} matrix.

You must also define a local coordinate system that is rotated 90 degrees about the y-axis. Then use this coordinate system n the piezoelectric material settings in order to rotate the material so that the poling direction is aligned with the x-axis.

MESHING

The thin central core that holds the piezoelectric actuator and the foam normally creates a relatively dense isotropic mesh. You can avoid this situation by scaling the mesh in the z direction with a factor of three. Doing so reduces the degrees of freedom from approximately 85,000 to 23,000 (by a factor of approximately 3.7) and hence it reduces both the solution time and memory requirements.

References

1. V. Piefort, *Finite Element Modelling of Piezoelectric Active Structures*, Ph.D. thesis, Université Libre de Bruxelles, Belgium, Dept. Mechanical Engineering and Robotics, 2001.

2. A. Benjeddou and others, *A Unified Beam Finite Element Model for Extension and Shear Piezoelectric Actuation Mechanisms*, CNAM (Paris, France), Structural Mechanics and Coupled Systems Laboratory, 1997.

Model Library path: `Structural_Mechanics_Module/Piezoelectric_Effects/shear_bender`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Start COMSOL Multiphysics. This invokes the **Model Navigator**, which you can also open from an already running COMSOL Multiphysics session by choosing **New** from the **File** menu.
2. On the **New** page select **3D** from the **Space dimension** list.
3. Click the **Multiphysics** button, then select **Structural Mechanics Module>Piezoelectric Effects>Piezo Solid>Static analysis** from the list of application modes.
4. Click **OK** to close the **Model Navigator**.

Note that this gives you second-order elements, **Lagrange - Quadratic**, by default.

GEOMETRY MODELING

Define the model geometry in draw mode:

1 Draw a block by first clicking the **Block** button on the Draw toolbar and then entering the following parameters in the **Block** dialog box:

LENGTH	EXPRESSION
X	0.1
Y	0.03
Z	0.018

2 Draw a second block by clicking the **Block** button and entering the following parameters:

LENGTH	EXPRESSION
X	0.1
Y	0.03
Z	0.002

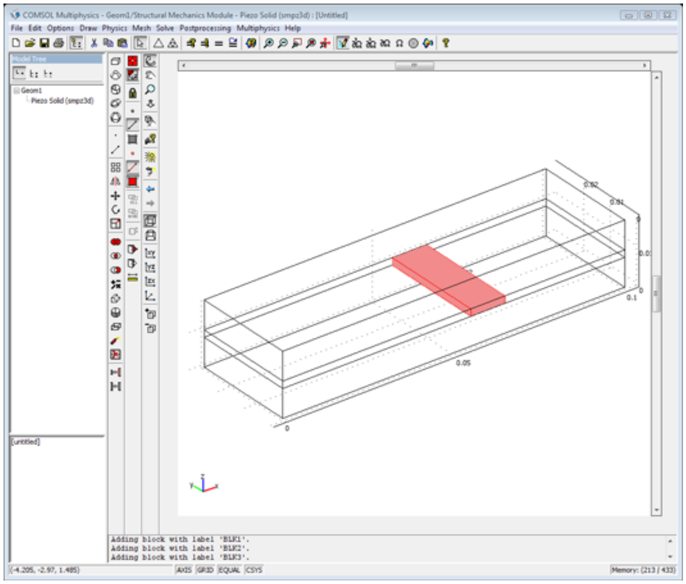
AXIS BASE POINT	EXPRESSION
x	0
y	0
z	0.008

3 Draw a third block by clicking the **Block** button and entering the following parameters:

LENGTH	EXPRESSION
X	0.01
Y	0.03
Z	0.002

AXIS BASE POINT	EXPRESSION
x	0.055
y	0
z	0.008

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



The basic geometry of the shear-actuated beam consisting of three blocks.

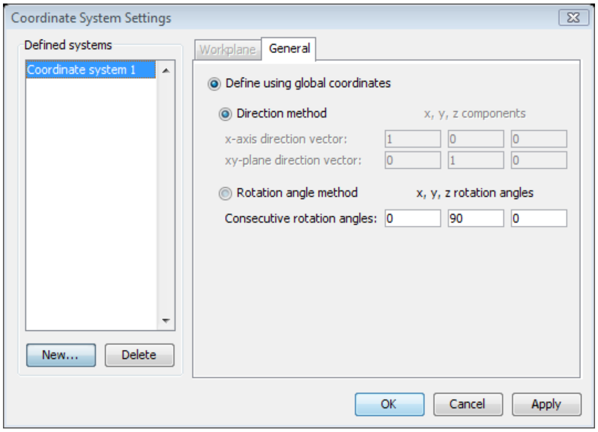
OPTIONS AND SETTINGS

1 From the **Options** menu choose **Coordinate Systems**.

2 In the **Coordinate System Settings** dialog box click the **New** button, then click **OK** in the **New Coordinate System** dialog box to use the default coordinate name.

3 Click the **General** tab. Click the **Define using global coordinates** option button, then click the **Rotation angle method** option button.

4 In the **x, y, z rotation angles** edit fields type `0`, `90`, and `0`, respectively, then click **OK** to close the dialog box.



Configuring the coordinate system.

The material properties for the foam and the aluminum are specified in the **Materials/ Coefficients Library** dialog box:

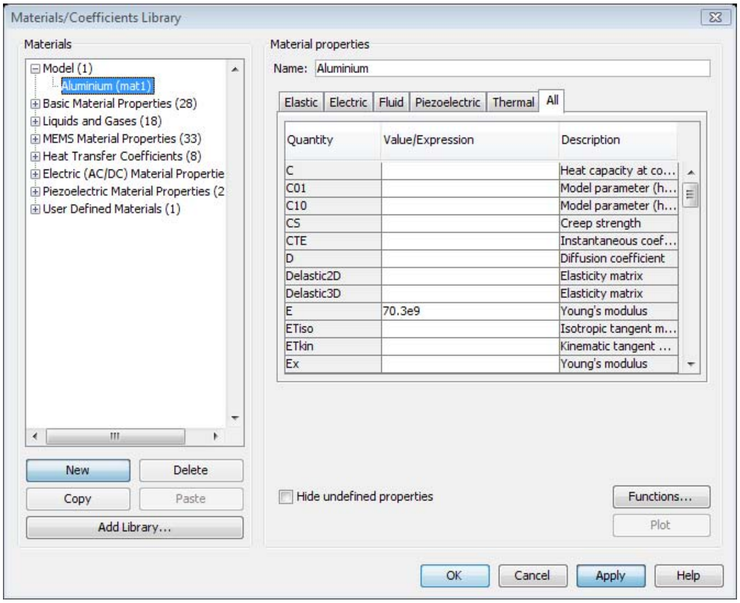
1 From the **Options** menu choose **Materials/Coefficients Library**.

2 Click the **New** button.

3 Change the **Name** to `Aluminum` and specify the following set of material parameters:

PARAMETER	EXPRESSION/VALUE	DESCRIPTION
`E`	`70e9`	Young's modulus
`nu`	`0.345`	Poisson's ratio
`rho`	`2690`	Density

4 Click **Apply**.

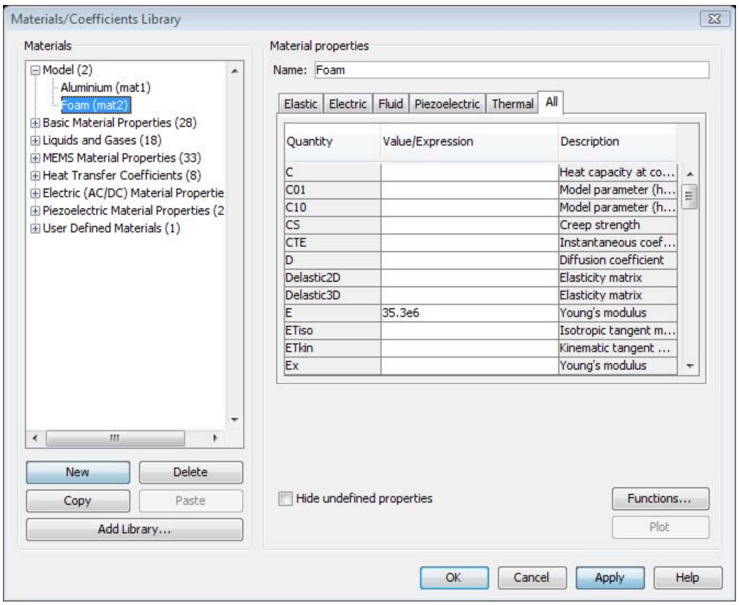


Material property values for aluminum.

5 Click the **New** button.

6 Change the **Name** to `Foam` and specify the following set of material parameters:

PARAMETER	EXPRESSION/VALUE	DESCRIPTION
`E`	`35.3e6`	Young's modulus
`nu`	`0.383`	Poisson's ratio
`rho`	`32`	Density



Material property values for foam.

7 Click **OK**.

PHYSICS SETTINGS

Subdomain Settings

1 Open the **Subdomain Settings** dialog box by choosing **Subdomain Settings** from the **Physics** menu. Go to the **Structural** page.

2 Select Subdomains 1 and 3 from the **Subdomain selection** list.

3 Select **Decoupled, isotropic** from the **Material model** list.

4 Select **Aluminum** from the **Library material** list; then click **Apply**.

5 Select Subdomains 2 and 5 from the **Subdomain selection** list.

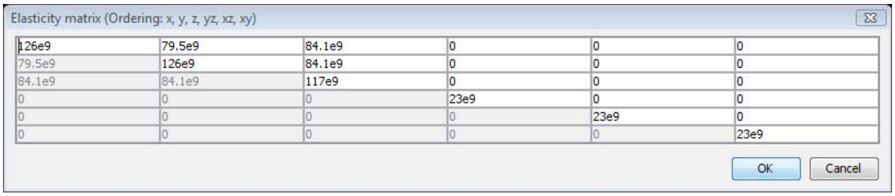
6 Select **Decoupled, isotropic** from the **Material model** list.

7 Select **Foam** from the **Library material** list; then click **Apply**.

8 Select Subdomain 4.

9 Select **Coordinate system 1** from the **Coordinate system** list.

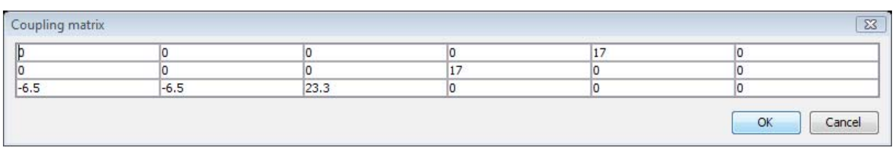
10 Click the **Edit** button associated with $\mathbf{c_E}$ and enter the following values into the **Elasticity matrix** dialog box; when complete, click **OK**.



126e9	79.5e9	84.1e9	0	0	0
	126e9	84.1e9	0	0	0
		117e9	0	0	0
			23e9	0	0
				23e9	0
					23e9

Entries for the elasticity matrix.

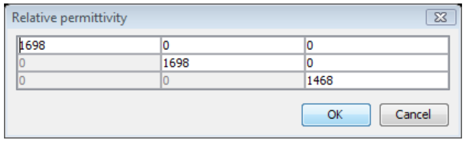
11 Click the **Edit** button associated with **e** and enter the following values into the **Coupling matrix, stress-charge form** dialog box; when complete, click **OK**.



0	0	0	0	17	0
0	0	0	17	0	0
-6.5	-6.5	23.3	0	0	0

Entries for the coupling matrix.

12 Click the **Edit** button associated with ε_{rs} and enter the following values into the **Relative permittivity matrix, stress-charge form** dialog box; when complete, click **OK**.



1698	0	0
	1698	
		1468

Entries for the relative permittivity matrix.

13 Click **OK**.

Boundary Conditions

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

2 Activate the **Interior boundaries** check box.

3 Fix the beam by selecting the following boundary condition on the **Constraint** page:.

SETTING	BOUNDARIES 1, 4, 7
Constraint condition	Fixed

4 Select Boundaries 14–19 from the **Boundary selection** list.

5 Go to the **Electric BC** page and select **Zero charge/Symmetry** from the **Boundary condition** list on the **Electric BC** page.

6 Select Boundary 16 and **Electric potential** from the **Boundary condition** list and enter 20 in the **Electric potential** edit field.

7 Select Boundary 17 and **Ground** from the **Boundary condition** list.

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

MESH GENERATION

1 From the **Mesh** menu choose **Free Mesh Parameters**.

2 Click the **Advanced** tab.

3 Type 3 in the **z-direction scale factor** edit field.

4 Click **Remesh**, then click **OK**.

POSTPROCESSING AND VISUALIZATION

Before computing the solution change the default plot to display a boundary plot using the deformed shape:

1 From the **Postprocessing** menu choose **Plot Parameters**.
2 On the **General** page clear the **Slice** check box, then select the **Boundary** check box and the **Deformed shape** check box.
3 Click the **Boundary** tab and select **Piezo Solid (smpz3d)>z-displacement** from the **Predefined quantities** list.
4 Click **OK**.

COMPUTING THE SOLUTION

To start computing the solution, click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION, CONTINUED

Figure 13-6 on page 543 shows the resulting plot.

Composite Piezoelectric Transducer

Introduction

This example shows how to set up a piezoelectric transducer problem following the work of Y. Kagawa and T. Yamabuchi (Ref. 1). The composite piezoelectric ultrasonic transducer has a cylindrical geometry that consists of a piezoceramic (NEPEC 6) layer, two aluminum layers, and two adhesive layers. The layers are organized as follows: aluminum layer—adhesive layer—piezoceramic layer—adhesive layer—aluminum layer.

The system applies an AC potential on the electrode surfaces of both sides of the piezoceramic layer. The potential in this example has a peak value of 1V in the frequency range 20 kHz to 106 kHz. The goal is to compute the susceptance (the imaginary part of the admittance) $Y = I/V$, where I is the total current and V is the potential, for a frequency range around the four lowest eigenfrequencies of the structure.

The first step finds the eigenmodes, and the second step runs a frequency sweep across an interval that encompasses the first four eigenfrequencies. Both analyses are fully coupled, and COMSOL Multiphysics assembles and solves both the electric and mechanical parts of the problem simultaneously.

The problem is axially symmetric, and you could analyze it using an axisymmetric application mode in 2D. However, in order to illustrate the modeling principles for more complicated problems, this example is in 3D.

When creating the model geometry, you make use of the symmetry by first making a cut along a midplane perpendicular to the central axis and then cutting out a 10 degree wedge. Doing so reduces memory requirements significantly.

Results

Figure 13-7 shows the input admittance in the transducer as a function of the excitation frequency.

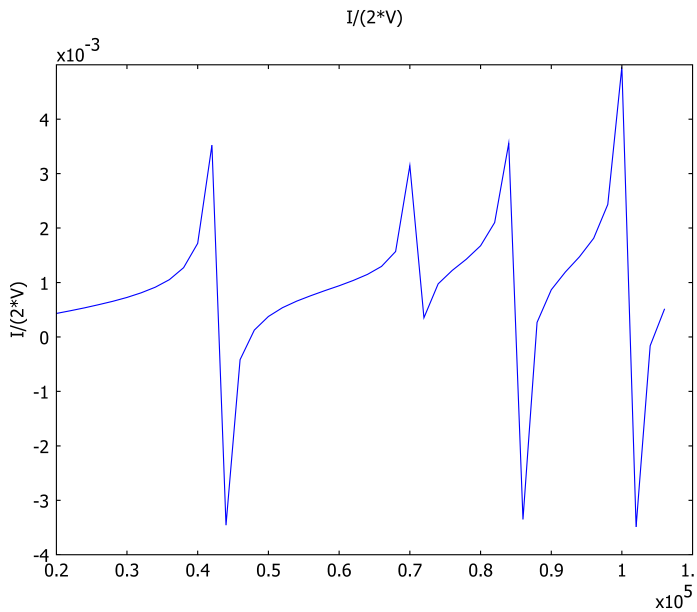


Figure 13-7: Input admittance as a function of excitation frequency

The result is in agreement with the work in Ref. 1. A small discrepancy close to the eigenfrequencies appears because the simulation uses no damping.

Reference

1. Y. Kagawa and T. Yamabuchi, "Finite Element Simulation of a Composite Piezoelectric Ultrasonic Transducer," *IEEE Transactions on Sonics and Ultrasonics*, vol. SU-26, no. 2, pp. 81–88, 1979.

Model Library path: `Structural_Mechanics_Module/Piezoelectric_Effects/composite_transducer`

MODEL NAVIGATOR

1 Open the **Model Navigator** and click the **New** tab. From the **Space dimension** list select **3D**.

2 In the list of application modes select **Structural Mechanics Module>Piezoelectric Effects>Piezo Solid>Eigenfrequency analysis**.

3 Click **OK**.

GEOMETRY MODELING

1 From the **Draw** menu choose **Work Plane Settings**.

2 This model uses the default work plane at $z = 0$, so simply click **OK**. Doing so creates the work plane and opens a 2D drawing area.

3 Click the **Ellipse/Circle (Centered)** button on the Draw toolbar. Use the right mouse button to create a circle centered on the origin and with a radius of approximately 1 (the exact size is not important).

4 Double-click the circle. In the **Radius** edit field enter `27.5e-3`. Click **OK**.

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

6 Click the **Line** button on the Draw toolbar. Draw a line from (0, 0) to (0.03, 0). Use the right mouse button to finish drawing lines.

7 Copy and paste the line onto itself with no displacements (either use Ctrl+C and Ctrl+V or use the **Edit** menu items).

8 Select the **Rotate** toolbar button from the Draw toolbar. In the **Rotation angle** edit field enter `10`. Click **OK**.

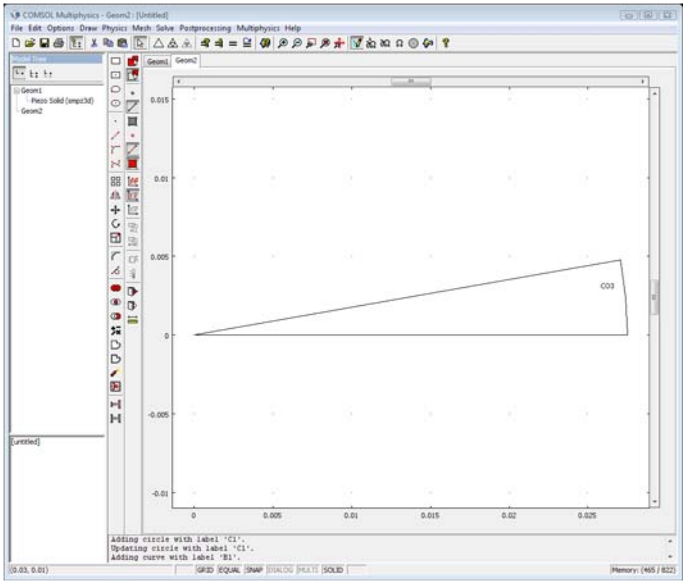
9 Select all objects (press Ctrl+A).

10 Click the **Coerce to Solid** button on the Draw toolbar.

11 Click the **Split Object** button on the Draw toolbar.

12 Select the larger object and press the Delete key.

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



Geometry for the composite piezoelectric transducer.

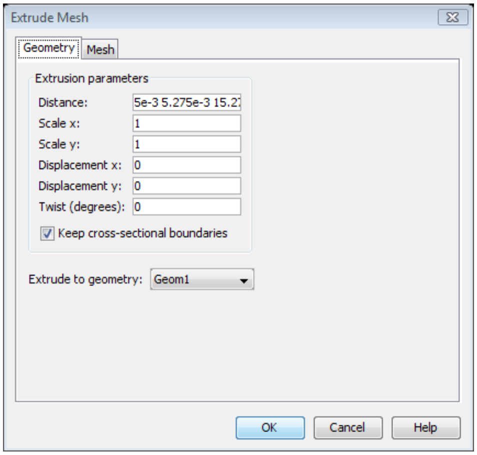
MESH GENERATION

This model uses prism elements by extruding a triangular mesh.

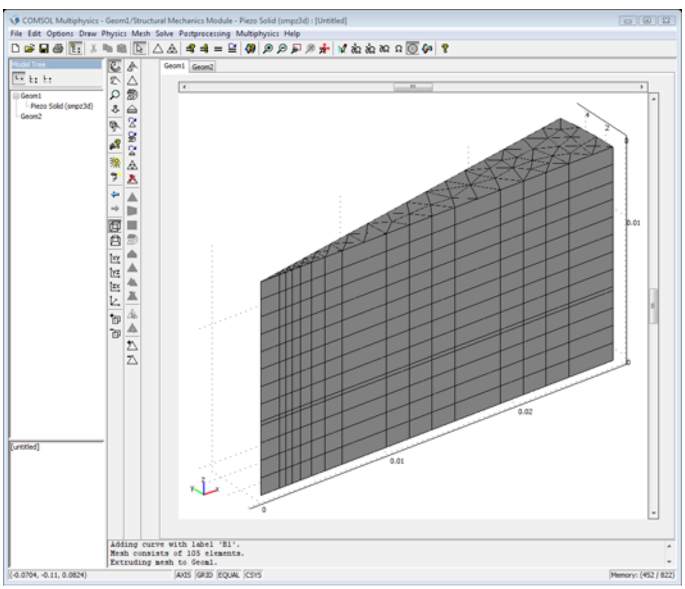
1 Click the **Initialize Mesh** button on the Main toolbar.

2 From the **Mesh** menu select **Extrude Mesh**.

3 In the **Distance** edit field enter `5e-3 5.275e-3 15.275e-3` (being careful to separate the entries with a space). Click **OK**. This step extrudes a mesh with three domains corresponding to the piezoceramic (Subdomain 1), the adhesive layer (Subdomain 2), and the aluminum layer (Subdomain 3).



Extruding the mesh to create three subdomains.



The finalized mesh with the three subdomains.

OPTIONS AND SETTINGS

1 From the **Options** menu select **Materials/Coefficients Library**.

2 Click **New**.

3 Change the **Name** to `Adhesive`.

4 Enter these material properties; when done, click **Apply**.

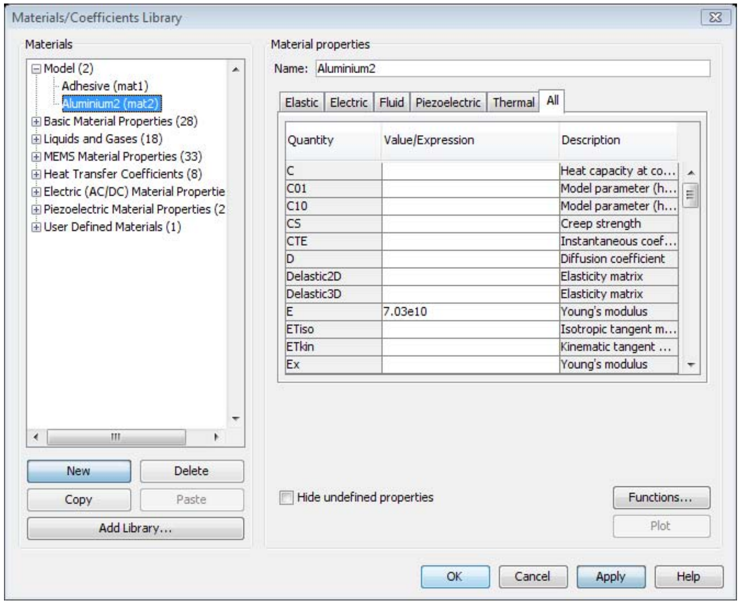
PARAMETER	EXPRESSION/VALUE	DESCRIPTION
E	1e10	Young's modulus
nu	0.38	Poisson's ratio
rho	1700	Density

5 Click the **New** button.

6 Change the name to `Aluminum2`.

7 Enter these material properties; when done, click **OK**.

PARAMETER	EXPRESSION/VALUE	DESCRIPTION
E	7.03e10	Young's modulus
nu	0.345	Poisson's ratio
rho	2690	Density



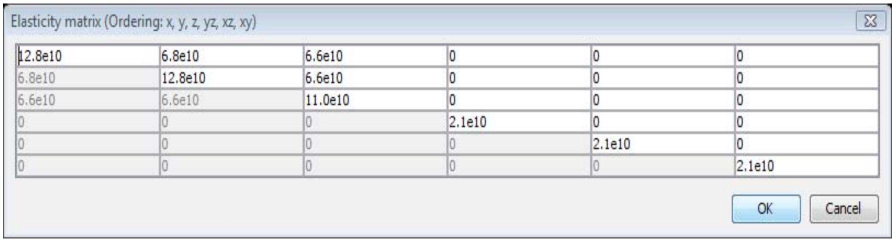
Material values to enter for the aluminum.

PHYSICS SETTINGS

Subdomain Settings

1 From the **Multiphysics** menu select **1 Geom1: Piezo Solid (smpz3d)**.
2 From the **Physics** menu select **Subdomain Settings** and go to the **Structural** page.
3 From the **Subdomain selection** list choose **1**.
4 See that the **Material Model** list has value **Piezoelectric**, then in the **Density** edit field enter 7730.

5 Click the **Edit** button associated with $\mathbf{c_E}$ and enter the following values into the **Elasticity matrix** dialog box; when finished, click **OK**:



12.8e10	6.8e10	6.6e10	0	0	0
	12.8e10	6.6e10	0	0	0
		11.0e10	0	0	0
			2.1e10	0	0
				2.1e10	0
					2.1e10

Entries for the elasticity matrix.

6 Click the **Edit** button associated with **e** and enter the following values into the **Coupling matrix, stress-charge form** dialog box; when finished, click **OK**:

0	0	0	0	0	0
0	0	0	0	0	0
-6.1	-6.1	15.7	0	0	0

Entries for the coupling matrix.

7 Click the **Edit** button associated with ε_{rS} and enter the following values into the **Relative permittivity, stress-charge form** dialog box; when finished, click **OK**:

993.53	0	0
	993.53	0
		993.53

Entries for the relative permittivity matrix.

8 In the **Subdomain selection** list choose Subdomains 2 and 3.

9 From the **Material Model** list select **Decoupled, isotropic**.

10 In the **Subdomain selection** list choose **2**.

11 Select **Adhesive** from the **Library material** list.

12 In the **Subdomain selection** list choose **3**.

13 Select **Aluminium2** from the **Library material** list.

14 Click **OK**.

Boundary Conditions

The total potential difference between the two electrodes (only one modeled here) is 1 V, but symmetry dictates that the potential is 0 V on the symmetry plane and 0.5 V on the electrode.

1 From the **Physics** menu choose **Boundary Settings**.

2 Activate the **Interior boundaries** check box.

3 In the **Boundary selection** list select all the boundaries (press Ctrl+A).

4 Go to the **Electric BC** page, find the **Boundary condition** list and select **Zero charge/ Symmetry**.

5 From the **Boundary selection** list choose Boundary 6.

6 In the **Boundary condition** list select **Electric potential**, and in the $\mathbf{V_0}$ edit field type `0.5`.

7 Select Boundary 3.

8 Change the boundary condition on the **Electric BC** page to **Ground**.

9 From the **Boundary selection** list choose Boundaries 1, 2, 3, 4, 5, 7, and 8.

10 Go to the **Constraint** page, and in the **Constraint condition** list select **Symmetry plane**.

The outer boundary is free to move, so you do not set any constraints.

11 Click **OK**.

COMPUTING THE EIGENFREQUENCY SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION OF THE EIGENFREQUENCIES

The lowest eigenfrequency is at approximately 43 kHz.

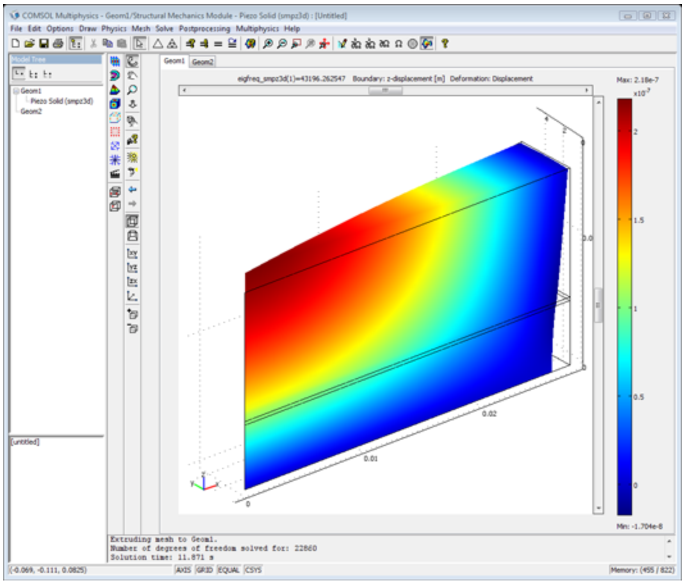
1 To visualize the solution of the lowest eigenfrequency, open the **Plot Parameters** dialog box from the **Postprocessing** menu.

2 On the **General** page, clear the **Slice** check box.

3 On the **Boundary** page, select the **Boundary plot** check box and select **Piezo Solid (smpz3d)>z-displacement (smpz3d)** in the **Predefined quantities** list.

4 On the **Deform** page, select the **Deformed shape plot** check box and see **Piezo Solid (smpz3d)>Displacement** in selected the **Predefined quantities** list.

5 Click **OK** to generate the visualization where you can see that the lowest eigenfrequency is at approximately 43 kHz.



Visualization of the lowest eigenfrequency for the piezoelectric transducer.

COMPUTING THE FREQUENCY RESPONSE SOLUTION

Next sweep over a frequency range from 20 kHz to 106 kHz in steps of 2 kHz.

1 From the **Physics** menu select **Properties**.

2 In the **Analysis type** list select **Frequency response**. Click **OK**.

3 From the **Solve** menu select **Solver Parameters**.

4 On the **General** page see that the **Parameter name** edit field shows `freq_smpz3d`.

5 In the **Parameter values** edit field enter `20e3:2e3:106e3`.

6 Click **OK**.

7 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION OF THE FREQUENCY RESPONSE

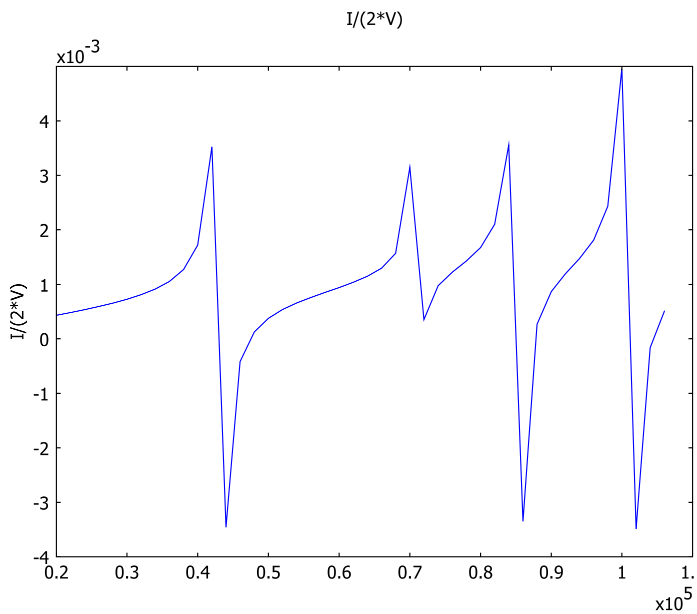
To compute the input admittance, use the current-density outflow variable, nJ, on the electrode surface. The total input current for the 10-degree wedge is then the surface integral of $-\mathrm{imag}(nJ)$ over one of the electrode surfaces. To obtain the total current for the entire structure, multiply the result by $360/10 = 36$.

1 From the **Options** menu select **Integration Coupling Variables>Boundary Variables**.
2 In the **Boundary selection** list choose Boundary 6 (corresponding to the electrode surface).
3 In the top row enter `I` as the name of the integration variable, then in the **Expression** column enter `-imag(nJ_smpz3d)*36`. Click **OK**.
4 From the **Solve** menu select **Update Model**.

The coupling variable `I` has global scope, and you can access it anywhere in the geometry model (the mesh). Use a domain plot to plot the value of Y versus frequency.

1 From the **Postprocessing** menu select **Domain Plot Parameters**.
2 Go to the **Point** page.
3 In the **Point selection** area choose a point on the electrode surface (for this exercise, select **2**).

4 In the **Expression** edit field enter `I/(2*V)`. Click **OK** to close the dialog box and plot the susceptance.



Plot of susceptance versus frequency for the piezoelectric transducer.

Recall that this example uses 0.5 V for the electrode potential but the overall potential difference is 1 V. Thus, to take symmetry into account, you must multiply the potential by 2.

Piezoresistive Elevator Button

Introduction

This is a simplified model of a design of a piezoresistive in-wall elevator button described in a patent application submitted by C.J. Slabinski and R.B. Leach (Ref. 5). The elevator button is designed with the following requirements in mind:

- durability with respect to high temperatures
- capability of producing signals that can be easily used in digital signal processing
- reliability, even when subjected to high loads

With respect to each of these requested characteristics, piezoresistive devices are superior to piezoelectrical ones. To make the device resistant against fire and heat, the button assembly includes parts of materials that are unresponsive to heat, such as clay or brick.

Piezoresistive materials change their resisitivity when subjected to mechanical stresses. This model concentrates on piezoresistivity in silicon-type materials, which is the most important group for practical applications. Although the effect is also observed in metals with crystal structures different from silicon, the piezoresistive effect is more than an order of magnitude higher in silicon than in metals (Ref. 1).

Note: This Model requires the Structural Mechanics module and the AC/DC Module.

Model Definition

Figure 13-8 shows the simplified elevator-button design that you use in this model.

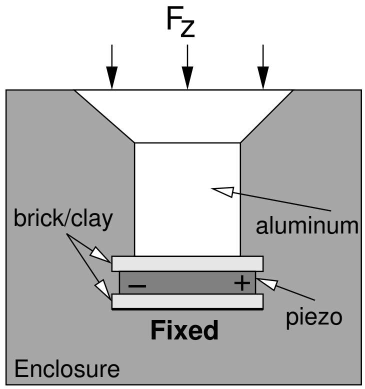


Figure 13-8: Schematic elevator button design.

The button is embedded in an enclosure formed by the elevator control panel. Its operation is modeled by a force, F_z, which acts on the button's upper boundary. The upper part of the button, composed of a conical and a cylindrical part, is made of aluminum. The conical shape ensures that the inner button parts are protected if the button is subjected to high impacts.

Furthermore, as mentioned in the introduction, it is vital that the button functions perfectly even at high ambient temperatures, such as in the event of fire. To shield the piezoresitive silicon layer, an adhesive, made of clay or brick, is attached to the resisitive layer's upper and lower sides.

To implement the button in COMSOL Multiphysics, you need two application modes:

- Solid, Stress-Strain, active in the whole geometry of the device
- Electric Currents, which is only active in the sensing resistor domain

To measure the change in the resistivity, a fixed voltage of 1 V is applied across the silicon layer. The integral of the current density over one of the resistor's contacts gives the ohmic resistance via Ohm's law, $U = RI$.

PIEZORESISTIVITY FOR AN ANISOTROPIC CRYSTAL

Below follows a short outline of the piezoresistive relations (Ref. 1). For a 3-dimensional anisotropic crystal, the electric field is related to the current-density field by a 3-by-3 resistivity tensor. In piezo crystals, the nine components always reduce to six values arranged in a symmetric tensor:

$$\begin{bmatrix} E_1 \\ E_1 \\ E_1 \end{bmatrix} = \begin{bmatrix} \rho_1 & \rho_6 & \rho_5 \\ \rho_6 & \rho_2 & \rho_4 \\ \rho_5 & \rho_4 & \rho_3 \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \\ I_3 \end{bmatrix} \tag{13-1}$$

For the cubic silicon lattice—with axes aligned with the <100> axes—ρ_1, ρ_2, and ρ_3 define the dependence of the electric field on the current along the same orthogonal directions; the other components are the cross terms.

The six resistivity components depend on the normal and shear stresses in the material (Ref. 2). Under stress-free conditions and with Cartesian coordinates aligned with the material <100> axes, the normal resistivity components are equal, and the cross terms are zero. Thus, under these conditions, the resistivity tensor is isotropic:

$$\rho_{3\text{-by-}3} = \rho_0 \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{13-2}$$

The relative changes in the resistivity can be written as a product of the structural stresses, σ, and the piezoresistive stress coefficients, Π. In Voigt notation, this reads:

$$\frac{1}{\rho_0}\begin{bmatrix} \Delta\rho_1 \\ \Delta\rho_2 \\ \Delta\rho_3 \\ \Delta\rho_4 \\ \Delta\rho_5 \\ \Delta\rho_6 \end{bmatrix} = \begin{bmatrix} \pi_{11} & \pi_{12} & \pi_{12} & 0 & 0 & 0 \\ \pi_{12} & \pi_{11} & \pi_{12} & 0 & 0 & 0 \\ \pi_{12} & \pi_{12} & \pi_{11} & 0 & 0 & 0 \\ 0 & 0 & 0 & \pi_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & \pi_{44} & 0 \\ 0 & 0 & 0 & 0 & 0 & \pi_{44} \end{bmatrix} \begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \tau_1 \\ \tau_2 \\ \tau_3 \end{bmatrix} \tag{13-3}$$

The following table lists the correspondences between the element indices or positions in Equation 13-3 and the tensor components along the spatial coordinate axes:

Π **MATRIX**		**RESISTIVITY**		**SHEAR STRESS**		Π **MATRIX**
RC	*xy*	*i*	*xy*	*i*	*xy*	**RC**
12	xy	ρ_6	ρ_{xy}	τ_3	τ_{xy}	66
13	xz	ρ_5	ρ_{xz}	τ_2	τ_{xz}	55
23	yz	ρ_4	ρ_{yz}	τ_1	τ_{yz}	44

Here, “R” and “C” stand for row and column position, respectively.

When modeling electric currents in conductive media, conductivity is used instead of resistivity. The anisotropic conductivity matrix corresponding to the resistivity matrix results from a simple inversion of the resistivity matrix.

Coordinate Transformations

To get a nice representation of the material parameters, it is common in piezoelectricity and piezoresistivity to rotate the coordinate system by 45° around the z-axis, so that the coordinate system aligns with the <110> directions for the crystal axes. The drawback of this representation is that the Cartesian axes (for which the stresses are defined) are aligned along the material <100> axes, in contrast to the representation used in structural mechanics (standard global coordinate system). Therefore, a local coordinate system is introduced for computing the resisitivity coefficients.

When you create and use a local coordinate system in COMSOL Multiphysics, you get an orthogonal 3-by-3 transformation matrix T, whose components appear as `coord`*n*`_T`*ij* variables in the equations. This matrix represents a transformation from local to global coordinates. The transformation of stresses from global to local coordinates is

$$\sigma_1 = T^{\mathrm{T}} \begin{bmatrix} \sigma_x & \sigma_{xy} & \sigma_{xz} \\ \sigma_{xy} & \sigma_y & \sigma_{yz} \\ \sigma_{xz} & \sigma_{yz} & \sigma_z \end{bmatrix} T$$

Use these stress components in the local representation to calculate the resistivity matrix, and then transform the resistivity matrix to global coordinates using the inverse transformation

$$\rho_{\mathrm{g}} = T \begin{bmatrix} \rho_1 & \rho_6 & \rho_5 \\ \rho_6 & \rho_2 & \rho_4 \\ \rho_5 & \rho_4 & \rho_3 \end{bmatrix} T^{\mathrm{T}} = T \begin{bmatrix} \rho_{11} & \rho_{12} & \rho_{13} \\ \rho_{21} & \rho_{22} & \rho_{23} \\ \rho_{31} & \rho_{32} & \rho_{33} \end{bmatrix} T^{\mathrm{T}}$$

Finally, invert this matrix and use the conductivity values in the conductive-media equation.

MATERIAL PROPERTIES

Table 13-1 lists the elastic properties for the materials used in the different button components. For aluminum and brick, you can use the basic material properties library included with COMSOL Multiphysics, whereas you enter the properties for the piezo element manually. The thermal expansion coefficients, α, are included in the table to simplify generalizing the model to take heat effects into account.

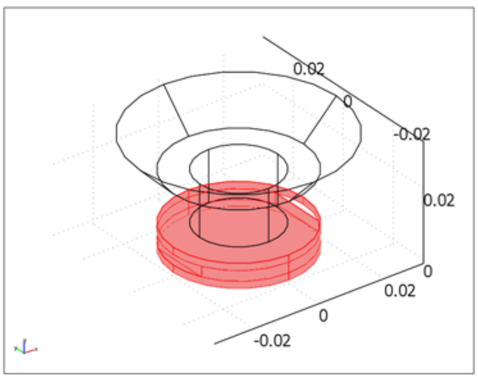
TABLE 13-1: ELASTIC MATERIAL PROPERTIES

PROPERTY	ALUMINUM	BRICK	PIEZO ELEMENT	DESCRIPTION
E (GPa)	70	17	169	Young's modulus
ν	0.33	0.3	0.278	Poisson's ratio
ρ (kg/m^3)	2700	2000	7850	Density
α (1/K)	$23 \cdot 10^{-6}$	$6 \cdot 10^{-6}$	$12 \cdot 10^{-6}$	Thermal expansion coefficient

BOUNDARY CONDITIONS

Solid, Stress-Strain—Constraints

Apply roller constraints to the boundaries highlighted in the figure below. Consider all other boundaries to be unconstrained.



Boundaries for which roller constraints apply.

Solid, Stress-Strain—Loads

On the top button surface, apply a homogeneous downward pressure of 1 kPa, corresponding to a total force of roughly 2.8 N.

Electric Currents

Apply a voltage of 1 V between the electrodes. All other boundaries are electrically insulating.

Results

Figure 13-9 displays the displacement of the boundary when full pressure is applied to the top button surface. The very small displacement values show that the button does not need to move for a signal to be generated.

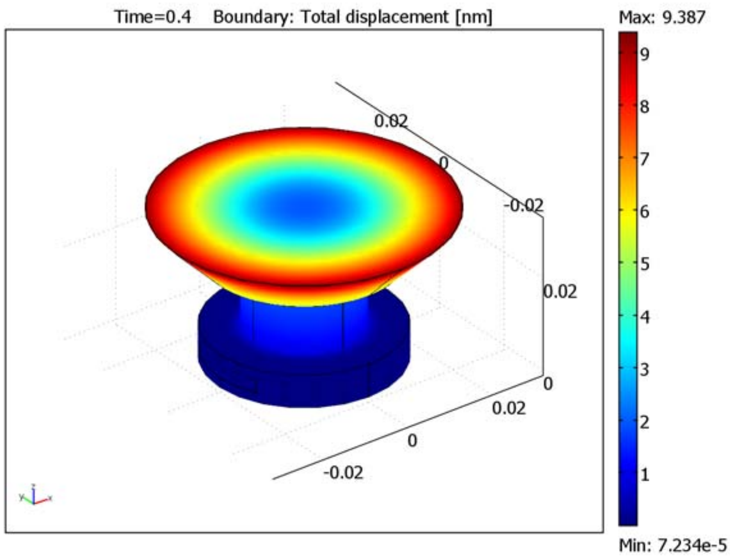


Figure 13-9: Total displacement of the button's boundary surfaces when a uniform downward pressure is applied on the top surface. The values of the displacement are so small that button movement is imperceptible.

Figure 13-10 shows the applied force and the current response as functions of time.

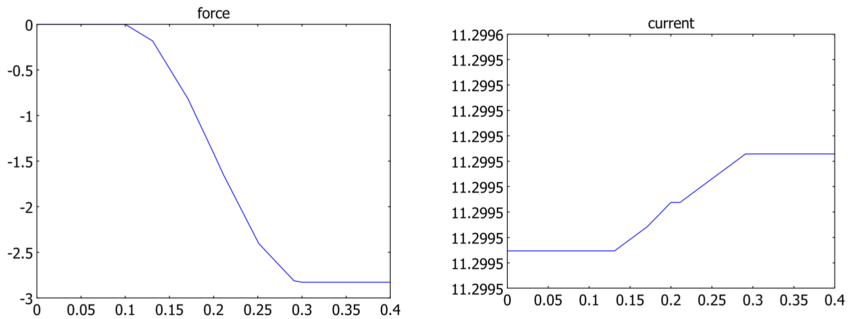


Figure 13-10: Applied force (left) and current response (right) as functions of time.

Because of the very small amplitude of the current signal, it is useful to plot $I - I_0$, where I_0 denotes the current in the absence of a load. Figure 13-11 shows this quantity as a function of time.

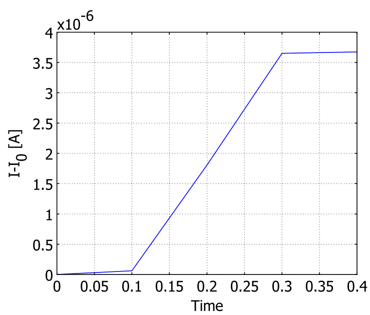


Figure 13-11: Differential current response.

From Figure 13-11 and the plot in the right panel of Figure 13-10 you can read off a signal-to-bias ratio of less than 10^{-6}. To allow the signal to be detected, the noise level needs to be even smaller than this. Thus, the simplified model of this example merely illustrates a possible application of piezoresistivity. The design must be refined and accompanied by the appropriate electronic circuitry to be technically viable.

References

1. *The MEMS Handbook*, ch. 16.4.2.5. Piezoresistivity in Silicon.

2. *The MEMS Handbook*, ch. 25.2. Piezoresistive Pressure Sensors.

3. COMSOL Model Database, id 724. Modeling Piezoresistivity.

4. C.S. Smith, “Piezoresistance Effect in Germanium and Silicon,” *Phys. Rev.*, vol. 94, pp. 42–49, 1954.

5. C.J. Slabinski and R.B. Leach, United States Patent, patent no. 5,040,640, patent date Aug. 20, 1991.

Model Library path: `Structural_Mechanics_Module/Piezoelectric_Effects/elevator_button`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 From the **Space dimension** list, select **3D**.

2 Select the application mode **Structural Mechanics Module>Solid, Stress-Strain>Quasi-static analysis**.

3 Click **Multiphysics**, then click **Add**.

4 Select the **AC/DC Module>Quasi-Statics, Electric>Electric Currents>Transient analysis** application mode

5 Click **Add**, then click **OK**.

OPTIONS AND SETTINGS

Constants

1 From the **Options** menu, choose **Constants**.

2 On separate rows in the dialog-box table, enter the resistivity and piezoresistive stress tensor components listed in the following table. Alternatively, click the **Import Variables From File** button, browse to the folder `Structural_Mechanics_Module/Piezoelectrical_Effects` in the Model Library root directory, select the file `elevator_button_const.txt`, and click **Open**.

NAME	EXPRESSION	DESCRIPTION
rho0	180e-6[ohm*m]	Resistivity of the unstressed material
p11	6.6e-11[1/Pa]	Piezoresistive stress coefficient component
p12	-1.1e-11[1/Pa]	Piezoresistive stress coefficient component

NAME	EXPRESSION	DESCRIPTION
p44	138.1e-11[1/Pa]	Piezoresistive stress coefficient component
P11, P22, P33	p11	Piezoresistive stress coefficient matrix component
P12, P13, P21, P23, P31, P32	p12	Piezoresistive stress coefficient matrix component
P44, P55, P66	p44	Piezoresistive stress coefficient matrix component

Coordinate Systems

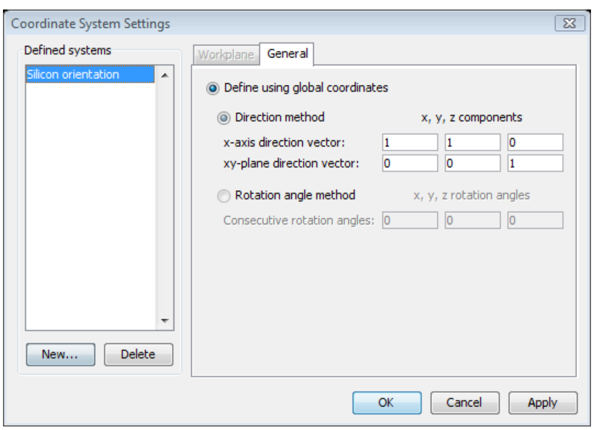
Because the piezoresistive coefficients are given in local coordinates, you must transform them to the standard global coordinate system used in the Solid, Stress-Strain application mode. Define the local coordinate system as follows:

1 From the **Options** menu, choose **Coordinate Systems**.

2 Click the **New** button. In the **Name** edit field, type `Silicon orientation`, then click **OK**.

3 Back in the **Coordinate System Settings** dialog box, enter the following settings on the **General** page:

SETTINGS	X	Y	Z
x-axis direction vector	1	1	0
xy-plane direction vector	0	0	1



4 Click **OK**.

Scalar Expressions

Next, add the stress-dependent resistivity and conductivity components in local and global coordinates defined in Table 13-2 through Table 13-5. You can either enter them manually in the **Scalar Expressions** dialog box or, more conveniently, load them directly from a data file included with the Structural Mechanics Module.

1 From the **Options** menu, choose **Expressions>Scalar Expressions**.

2 Enter the stress, resistivity, and conductivity components listed in the following tables or (preferably) click the **Import Variables From File** button, browse to the folder `Structural_Mechanics_Module/Piezoelectrical_Effects` in the Model Library root directory, select the file `elevator_button_expr.txt`, and click **Open**.

3 Click **OK** to close the **Scalar Expressions** dialog box.

TABLE 13-2: MATERIAL STRESS IN LOCAL COORDINATES

NAME	EXPRESSION
sxl	sxl_smsld
syl	syl_smsld
szl	szl_smsld
sxzl	sxzl_smsld
syzl	syzl_smsld
sxyl	sxyl_smsld

TABLE 13-3: RELATIVE RESISTIVITY (RHO/RHO0) IN LOCAL COORDINATES

NAME	EXPRESSION
rhol11	1+P11*sxl+P12*syl+P13*szl
rhol22	1+P21*sxl+P22*syl+P23*szl
rhol33	1+P31*sxl+P32*syl+P33*szl
rhol23	P44*syzl
rhol13	P55*sxzl
rhol12	P66*sxyl

TABLE 13-4: RELATIVE RESISTIVITY IN GLOBAL COORDINATES

NAME	EXPRESSION
rho11	coord1_T11*(rhol11*coord1_T11+rhol12*coord1_T12+rhol13*coord1_T13)+coord1_T12*(rhol12*coord1_T11+rhol22*coord1_T12+rhol23*coord1_T13)+coord1_T13*(rhol13*coord1_T11+rhol23*coord1_T12+rhol33*coord1_T13)
rho22	coord1_T21*(rhol11*coord1_T21+rhol12*coord1_T22+rhol13*coord1_T23)+coord1_T22*(rhol12*coord1_T21+rhol22*coord1_T22+rhol23*coord1_T23)+coord1_T23*(rhol13*coord1_T21+rhol23*coord1_T22+rhol33*coord1_T23)
rho33	coord1_T31*(rhol11*coord1_T31+rhol12*coord1_T32+rhol13*coord1_T33)+coord1_T32*(rhol12*coord1_T31+rhol22*coord1_T32+rhol23*coord1_T33)+coord1_T33*(rhol13*coord1_T31+rhol23*coord1_T32+rhol33*coord1_T33)
rho12	coord1_T11*(rhol11*coord1_T21+rhol12*coord1_T22+rhol13*coord1_T23)+coord1_T12*(rhol12*coord1_T21+rhol22*coord1_T22+rhol23*coord1_T23)+coord1_T13*(rhol13*coord1_T21+rhol23*coord1_T22+rhol33*coord1_T23)
rho23	coord1_T21*(rhol11*coord1_T31+rhol12*coord1_T32+rhol13*coord1_T33)+coord1_T22*(rhol12*coord1_T31+rhol22*coord1_T32+rhol23*coord1_T33)+coord1_T23*(rhol13*coord1_T31+rhol23*coord1_T32+rhol33*coord1_T33)
rho13	coord1_T11*(rhol11*coord1_T31+rhol12*coord1_T32+rhol13*coord1_T33)+coord1_T12*(rhol12*coord1_T31+rhol22*coord1_T32+rhol23*coord1_T33)+coord1_T13*(rhol13*coord1_T31+rhol23*coord1_T32+rhol33*coord1_T33)

TABLE 13-5: CONDUCTIVITY

NAME	EXPRESSION
det_rho	rho11*rho22*rho33+2*rho12*rho23*rho13-rho13^2*rho22-rho12^2*rho33-rho11*rho23^2
sigma11	(rho22*rho33-rho23^2)/(rho0*det_rho)
sigma22	(rho11*rho33-rho13^2)/(rho0*det_rho)
sigma33	(rho11*rho22-rho12^2)/(rho0*det_rho)
sigma12	(rho13*rho23-rho12*rho33)/(rho0*det_rho)
sigma13	(rho12*rho23-rho13*rho22)/(rho0*det_rho)
sigma23	(rho12*rho13-rho11*rho23)/(rho0*det_rho)

GEOMETRY MODELING

1 Click the **Cylinder** button on the Draw toolbar. Create a cylinder with **Radius** `0.02`, **Height** `0.002`, and the **Axis base point** at (`0`, `0`, `0`). Click **OK** to close the dialog box.

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

3 Repeat this procedure for three more cylinders with the following values:

NAME	RADIUS	HEIGHT	AXIS BASE POINT
CYL2	`2e-2`	`3e-3`	`(0,0,2e-3)`
CYL3	`2e-2`	`2e-3`	`(0,0,5e-3)`
CYL4	`12e-3`	`15e-3`	`(0,0,7e-3)`

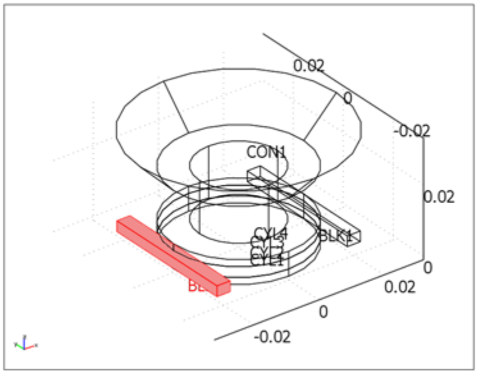
4 Click the **Cone** button on the Draw toolbar and specify the following parameters:

PARAMETER	VALUE
Axis base point	`(0,0,3.2e-2)`
Radius	`3e-2`
Height	`1e-2`
Semi-angle	`45`
Axis direction vector	`(0,0,-1)`

5 Click the **Block** button on the Draw toolbar. In the **Base** area, click **Center**. Set the **Axis base point** to (`0.02`, `0`, `0.0035`) and the **Length** to (`0.004`, `0.04`, `0.003`). Click **OK** to close the dialog box and create the block BLK1.

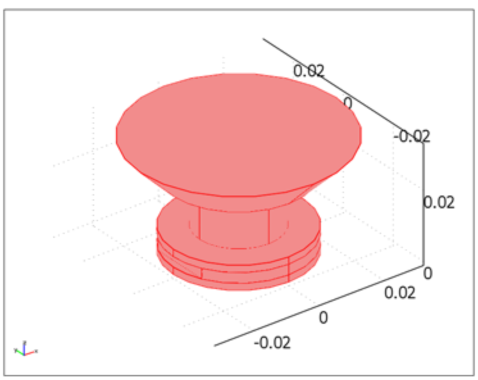
6 Select BLK1 and create a second block, BLK2, by pressing first Ctrl+C and then Ctrl+V. In the **Paste** dialog box, to specify the **Displacements** (`-0.04`, `0`, `0`). Click **OK**.

The geometry in the drawing area should now look like that in the following figure.



7 From the **Draw** menu, choose **Create Composite Object**. In the **Set formula** edit field, enter the expression `CYL1+CYL2+CYL3+CYL4+CON1-BLK1-BLK2`. Click **OK**.

The geometry is now complete and should look like that in the figure below.



PHYSICS SETTINGS

Subdomain Settings—Solid, Stress-Strain

1 From the **Multiphysics** menu, select **1 Solid, Stress-Strain (smsld)**.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select Subdomains 1 and 5, then click the **Load** button in the **Material settings** area.

4 From the **Basic Material Properties** library, select **Aluminum**, then click **OK**.

5 Select Subdomains 2 and 3, then click the **Load** button.

6 From the **Basic Material Properties** library, select **Brick**, then click **OK**.

7 Select Subdomain 4 and specify settings according to the following table:

PROPERTY	VALUE
Coordinate system	Silicon orientation
E	`169e9`
ν	`0.278`
α	`12e-6`
ρ	`7850`

8 Click **OK**.

Boundary Conditions—Solid, Stress-Strain

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

2 On the **Constraint** page, specify the following constraint condition:

SETTINGS	BOUNDARIES 4–6, 8, 9, 13, 14, 17, 23–25, 28–30, 32
Constraint condition	Roller

The boundaries not listed above are free, which is the default constraint condition.

3 Go to the **Load** page and select Boundary 3.

4 In the $\mathbf{F_z}$ edit field, type `-1[kPa]*flc1hs(t[1/s]-0.2,0.1)`.

5 Click **OK**.

Subdomain Settings—Electric Currents

1 From the **Multiphysics** menu, choose **2 Electric Currents (emqvw)**.

2 Select Subdomains 1–3 and 5. Clear the **Active in this domain** check box.

3 Select Subdomain 4. In the σ edit field type `sigma11 sigma12 sigma22 sigma13 sigma23 sigma33`. The editable pop-up window displays the components' positions in the anisotropic, symmetric electric conductivity tensor.

4 Click **OK**.

Boundary Conditions—Electric Currents

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

2 Select the **Interior boundaries** check box.

3 Select all boundaries and set the boundary condition to **Electric insulation**.

4 Select Boundary 13. From the **Boundary condition** list, choose **Electric potential**. In the **V_0** edit field, type `1`.

5 Select Boundary 32. From the **Boundary condition** list, choose **Ground**.

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

Integration Coupling Variables

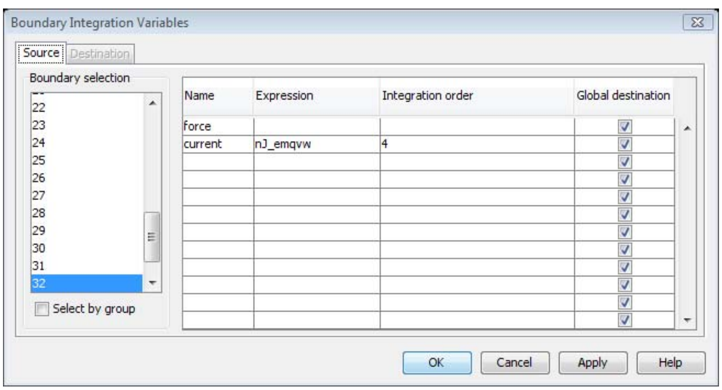
To have a probe plot for the developing of the electric current and the applied load on the top surface, define two boundary integration variables.

1 From the **Options** menu, select **Integration Coupling Variables>Boundary Variables**.

2 Define the following integration variables by first selecting the boundary, and then entering the name and expression on the first available row in the dialog box table.

BOUNDARY	NAME	EXPRESSION
3	`force`	`Fzg_smsld`
32	`current`	`nJ_emqvw`

In the **Integration order** and **Global destination** columns, leave the default settings.



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

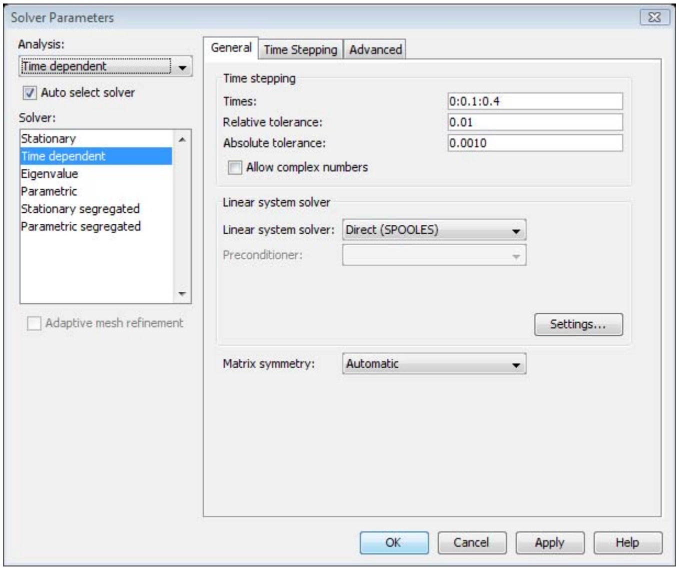
MESH GENERATION

Click the **Initialize Mesh** button on the Main toolbar to generate a default mesh.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 On the **General** page, type `0:0.1:0.4` in the **Times** edit field. Also, verify that the **Linear system solver** is set to **Direct (SPOOLES)**.



3 From the **Postprocessing** menu, choose **Probe Plot Parameters**.

4 In the **Probe Plot Parameters** dialog box, click the **New** button.

5 From the **Plot type** list, select **global**. In the **Plot name** edit field, type `force`.

6 Click **OK**.

7 In the **Expression to plot** area, the integration coupling variable `force` in the **Expression** edit field.

8 In the same fashion, define a probe plot with the label **current** for the integration coupling variable with the same name.

9 Click **OK** to close the **Probe Plot Parameters** dialog box.

10 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To reproduce the plot of the boundary displacement field displayed in Figure 13-9, follow these steps:

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

2 On the **General** page, clear the **Slice** check box and select the **Boundary** check box in the **Plot type** area.

3. Click the **Boundary** tab and verify that **disp_smsld** is selected from the **Predefined quantities** list. From the **Unit** list, select **nm**.
4. Click **OK** to close the dialog box and generate the plot.
5. Click the **Scene Light** button on the Camera toolbar.

Reproduce the plot in Figure 13-11 with these instructions:

1. From the **Postprocessing** menu, choose **Global Variables Plot**.
2. In the **Expression** edit field, type current-with(1,current).
3. Click the **Add Entered Expression** button.
4. Click the **Axis/Title** button.
5. Click the right option buttons for **Title** and **Second axis label**.
6. In the **Second axis label** edit field, type I-I₀ [A].
7. From the **Plot in** list, select **New figure**.
8. Click **OK**.

SAW Gas Sensor

Introduction

A surface acoustic wave (SAW) is an acoustic wave propagating along the surface of a solid material. Its amplitude decays rapidly, often exponentially, with the depth of the material. SAWs are featured in many kinds of electronic components, including filters, oscillators, and sensors. SAW devices typically use electrodes on a piezoelectric material to convert an electric signal to a SAW, and back again.

In this model, you investigate the resonance frequencies of a SAW gas sensor. The sensor consists of an interdigitated transducer (IDT) etched onto a piezoelectric $LiNbO_3$ (lithium niobate) substrate and covered with a thin polyisobutylene (PIB) film. The mass of the PIB film increases as PIB selectively adsorbs CH_2Cl_2 (dichloromethane, DCM) in air. This causes a shift in resonance to a slightly lower frequency.

Model Definition

Figure 13-12 shows a conceptual view of the gas sensor in this model. IDTs used in SAW devices may have hundreds of identical electrodes, and each electrode can be about 100 times longer than it is wide. You can therefore neglect the edge effects and reduce the model geometry to the periodic unit cell shown in Figure 13-13. The height of this cell does not have to extend all the way to the bottom of the substrate but only a few wavelengths down, so that the SAW has almost died out at the lower boundary. In the model, this boundary is fixed to a zero displacement.

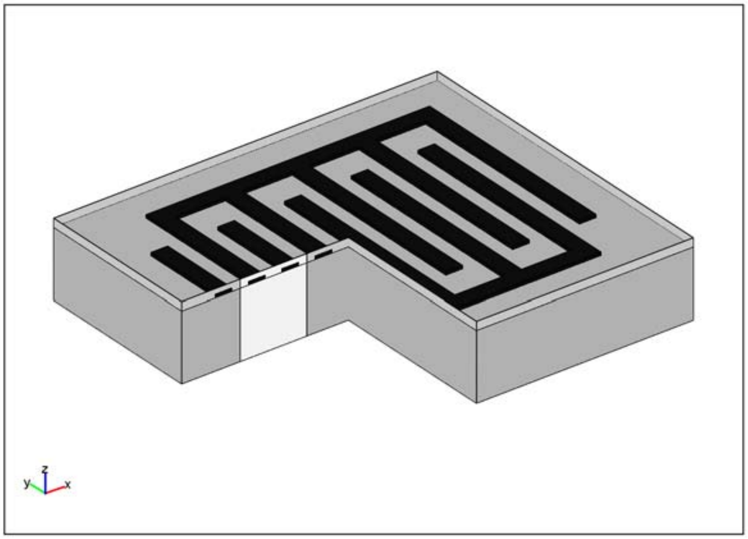


Figure 13-12: Conceptual view of the SAW gas sensor, showing the IDT electrodes (in black), the thin PIB film (light gray), and the $LiNbO_3$ substrate (dark gray). For the sake of clarity, the dimensions are not to scale and the IDT has fewer electrodes than in common devices. A slice of the geometry is removed to reveal the modeled unit cell (in white).

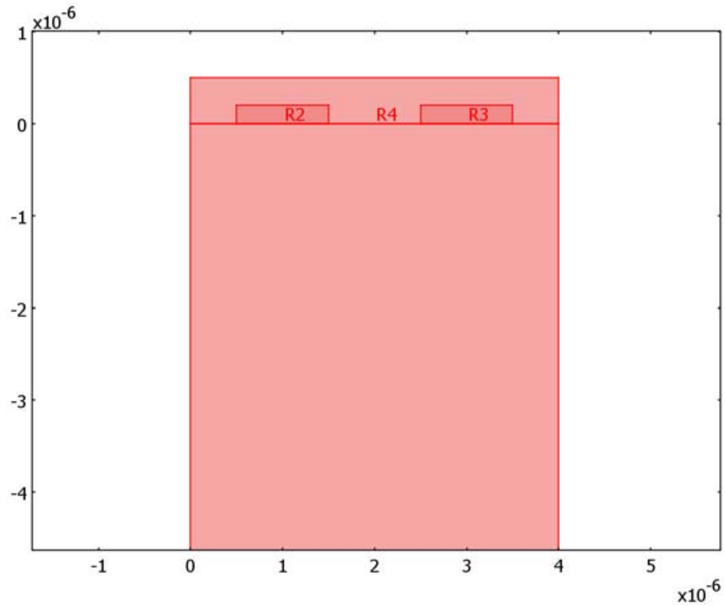


Figure 13-13: The modeled geometry of the model. A 500 nm PIB film covers two 1 µm-wide electrodes on top of the $LiNbO_3$ substrate. The substrate subdomain continues below the lower frame of the picture and has a total height of 22 µm. In the first version of the model, the substrate is the only active subdomain.

You set up the model in the Piezo Plane Strain application mode, which requires the out-of-plane strain component to be zero. This should be a valid assumption, considering that the SAW is generated in the model plane and that the sensor is thick in the out-of-plane direction.

The first version of the model deals only with free SAW propagation in the $LiNbO_3$ substrate, without any applied electric field. In order to find the velocity of the wave, we use periodic boundary conditions to dictate that the voltage and the displacements be the same along both vertical boundaries of the geometry. This implies that the wavelength will be an integer fraction of the width of the geometry. The lowest SAW eigenmode has its wavelength equal to the width of the geometry, 4 µm. The eigenfrequency of this mode multiplied by 4 µm hence gives the velocity of the wave.

In a second version of the model, the aluminum IDT electrodes and the PIB film are added. This causes the lowest SAW mode to split up in two eigensolutions, the lowest one representing a series resonance, where propagating waves interfere constructively and the other one a parallel ("anti-") resonance, where they interfere destructively. These two frequencies constitute the edges of the stopband, within which no waves can propagate through the IDT.

The adsorption of DCM gas is represented as a slight increase of the density of the PIB film. In the third and final version of the model, the sensor is exposed to 100 ppm of DCM in air at atmospheric pressure and room temperature. The "partial density" of DCM in the PIB film is then calculated as

$$\rho_{\mathrm{DCM,PIB}} = KMc\,,$$

where $K = 10^{1.4821}$ (Ref. 1) is the air/PIB partition coefficient for DCM, M is its molar mass, and

$$c = 100 \cdot 10^{-6} \cdot p/(RT)$$

is its concentration in air.

The substrate used in the simulation is YZ-cut $LiNbO_3$ with properties cited in Ref. 2. The density of the PIB film is from Ref. 1. The Poisson's ratio is taken to be 0.48, which corresponds to a rather rubbery material. The Young's modulus is set to 10 GPa. Even at the comparatively high frequencies considered in this model, this is likely an overestimation. However, a much lower value would result in a multitude of eigenmodes located inside the film. While those may be important to consider in designing a SAW sensor, the focus in this model is on the SAW modes. Also, any effects of the DCM adsorption on other material properties than the density are neglected.

Results

Figure 13-7 shows the SAW as it propagates along the surface of the piezoelectric substrate. The frequency corresponding to a 4 µm wavelength computes to 870 MHz, giving a phase velocity of 3479 m/s.

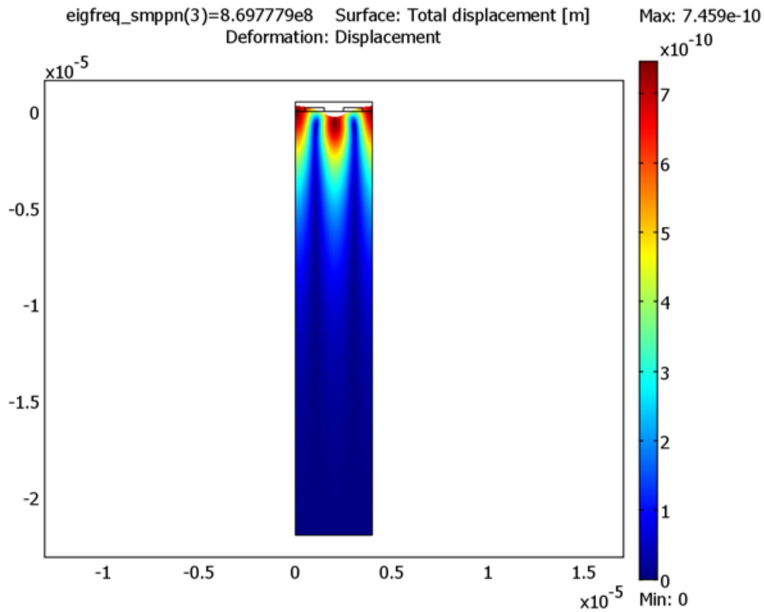


Figure 13-14: Deformed shape plot of a freely propagating SAW in the substrate. The color scale shows the magnitude of the displacements.

In the full model with the periodic IDT and the thin film included, the resonance and anti-resonance frequencies evaluate to 841 MHz and 850 MHz, respectively. Figure 13-15 and Figure 13-16 show the electric potential distribution characteristics for these solutions.

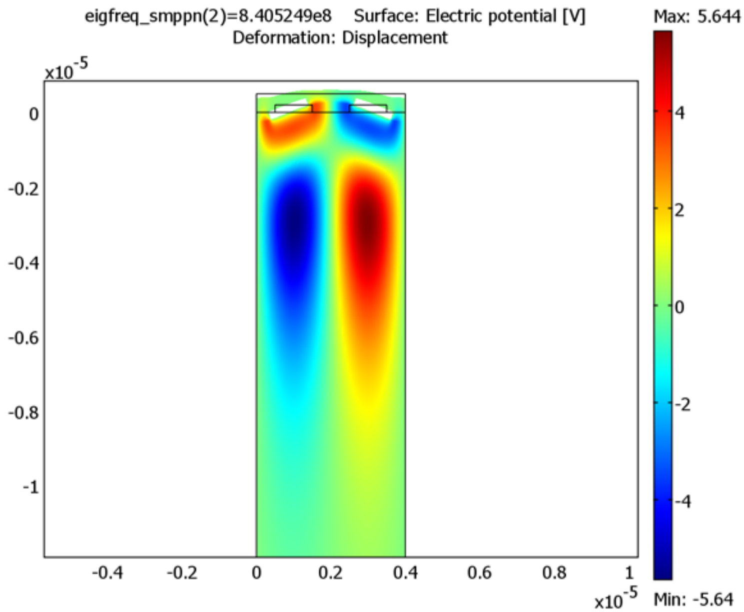


Figure 13-15: Electric potential distribution and deformations at resonance, 841 MHz. The potential is symmetric with respect to the center of each electrode.

Exposing the sensor to a 100 ppm concentration of DCM in air leads to a resonance frequency shift of 227 Hz downwards. This is computed by evaluating the resonance frequency before and after increasing the density of adsorbed DCM to that of the PIB domain.

Note that the computational mesh is identical in both these solutions. This implies that the relative error of the frequency shift is similar to that of the resonance frequency itself. Thus the shift is accurately evaluated despite being a few magnitudes smaller than the absolute error of the resonance frequency.

In a real setup, the drift is often measured by mixing the signal from a sensor exposed to a gas with a reference signal from one protected from the gas. The beat frequency then gives the shift.

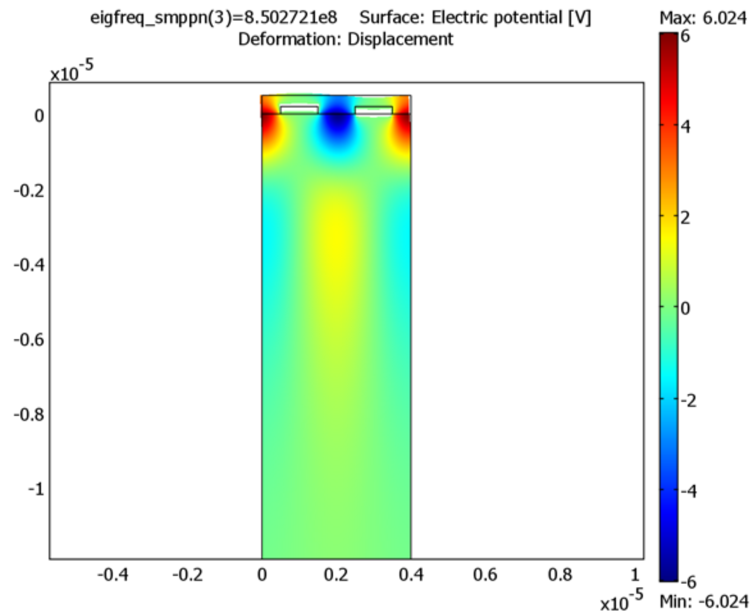


Figure 13-16: Electric potential distribution and deformations at antiresonance, 851 MHz. The potential is antisymmetric with respect to the center of the electrodes.

References

1. K. Ho and others, "Development of a Surface Acoustic Wave Sensor for In-Situ Monitoring of Volatile Organic Compounds", *Sensors* vol. 3, pp. 236–247, 2003.

2. Ahmadi and others, "Characterization of multi- and single-layer structure SAW sensor [gas sensor]", *Sensors 2004, Proceedings of IEEE*, vol. 3, pp. 1129–1132, 2004.

Model Library path: `Structural_Mechanics_Module/Piezoelectric_Effects/SAW_gas_sensor`

MODEL NAVIGATOR

1 Open the **Model Navigator** and click the **New** tab.

2 From the **Space dimension** list select **2D**.

3 In the list of application modes select **Structural Mechanics Module>Piezoelectric Effects>Piezo Plane Strain>Eigenfrequency analysis**.

4 Click **OK**.

GEOMETRY MODELING

1 Create the following rectangles by repeatedly using **Draw>Specify Objects>Rectangle**:

WIDTH	HEIGHT	BASES: CORNER X	BASE: CORNER Y
4	22	0	-22
1	0.2	0.5	0
1	0.2	2.5	0
4	0.5	0	0

2 Select all objects and choose **Draw>Modify>Scale**. In the dialog box that appears, enter `1e-6` for both scale factors; then click **OK**.

3 Click the **Zoom Extents** button on the Main toolbar to zoom in on the now micron-sized geometry.

OPTIONS AND SETTINGS

1 Choose **Options>Constants**.

2 Define the following constant names, expressions, and (optionally) descriptions:

NAME	EXPRESSION	DESCRIPTION
`p`	`101.325[kPa]`	`Air pressure`
`T`	`25[degC]`	`Air temperature`
`R`	`8.3145[Pa*m^3/(K*mol)]`	`Gas constant`
`c_DCM_air`	`100e-6*p/(R*T)`	`DCM concentration in air`
`M_DCM`	`84.93[g/mol]`	`Molar mass of DCM`
`K`	`10^1.4821`	`PIB/air partition constant for DCM`
`rho_DCM_PIB`	`K*M_DCM*c_DCM_air`	`Mass concentration of DCM in PIB`

NAME	EXPRESSION	DESCRIPTION
rho_PIB	0.918[g/cm^3]	Density of PIB
E_PIB	10[GPa]	Young's modulus of PIB
nu_PIB	0.48	Poisson's ratio of PIB
eps_PIB	2.2	Relative permittivity of PIB

3 Click **OK**.

PHYSICS SETTINGS

In the first version of the model, you compute the velocity for SAW propagation in a homogenous, electrically insulated $LiNbO_3$ substrate. The supplied material data are with reference to the *xy*-plane.

Subdomain Settings

1 From the **Physics** menu, open the **Subdomain Settings** dialog box.
2 Select Subdomains 2–4 and clear the **Active in this domain** check box.
3 Select Subdomain 1 and select **Material orientation: xy plane**.
4 Click the **Edit** button associated with $\mathbf{c_E}$ and enter the following values into the **Elasticity matrix** dialog box; when finished, click **OK**.

$$\begin{bmatrix} 2.424e11 & 0.752e11 & 0.752e11 & 0 & 0 & 0 \\ & 2.03e11 & 0.573e11 & 0 & 0.085e11 & 0 \\ & & 2.03e11 & 0 & -0.085e11 & 0 \\ & & & 0.752e11 & 0 & 0.085e11 \\ & & & & 0.595e11 & 0 \\ & & & & & 0.595e11 \end{bmatrix}$$

5 Click the **Edit** button associated with **e** and enter the following values into the **Coupling matrix** dialog box; when finished, click **OK**.

$$\begin{bmatrix} 1.33 & 0.23 & 0.23 & 0 & 0 & 0 \\ 0 & 0 & 0 & -2.5 & 0 & 3.7 \\ 0 & -2.5 & 2.5 & 0 & 3.7 & 0 \end{bmatrix}$$

6 Click the **Edit** button associated with ε_{rS} and enter the following values into the **Relative permittivity** dialog box; when finished, click **OK**.

$$\begin{bmatrix} 28.7 & 0 & 0 \\ & 85.2 & 0 \\ & & 85.2 \end{bmatrix}$$

7 Enter `4647` in the **Density** edit field.

8 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

1 From the **Physics** menu choose **Boundary Settings**.

2 Select Boundary 2 and set the **Constraint condition** to **Fixed**.

3 Select all exterior boundaries (1, 2, 4, 7, 10, 12, 15, 16).

4 On the **Electric BC** page, set the **Boundary condition** to **Zero charge/Symmetry**.

5 Click **OK**.

6 Choose **Physics>Periodic Conditions>Periodic Boundary Conditions**.

7 On the **Source** tab, select Boundary 1 and enter `u` in the first **Expression** edit field.

8 On the **Destination** page, check Boundary 16 and enter `u` in the **Expression** edit field.

9 On the **Source Vertices** page, select Vertex 1 and click the right double-arrow. Then select Vertex 2 and click the right double-arrow.

10 On the **Destination Vertices** page, select Vertex 12 and click the right double-arrow, then Vertex 13 and the right double-arrow.

11 Define the expressions `v` and `V` in a similar fashion, starting by entering them in the **Expression** edit field on the **Source** page, on rows 2 and 3 respectively.

12 Click **OK** to close the **Periodic Boundary Conditions** dialog box.

MESH GENERATION

1 Choose **Mesh>Free Mesh Parameters**.

2 From the **Predefined mesh sizes** list, choose **Extremely fine**.

3 On the **Subdomain** page, select all subdomains and set the **Method** to **Quad**.

4 On the **Boundary** page, select the upper boundaries of the substrate (4, 7, 10, 12, 15) and enter `0.05e-6` for the **Maximum element size**.

5 Click **Remesh**, then click **OK**. When done, a zoom-in on the upper part of the geometry should look like Figure 13-17.

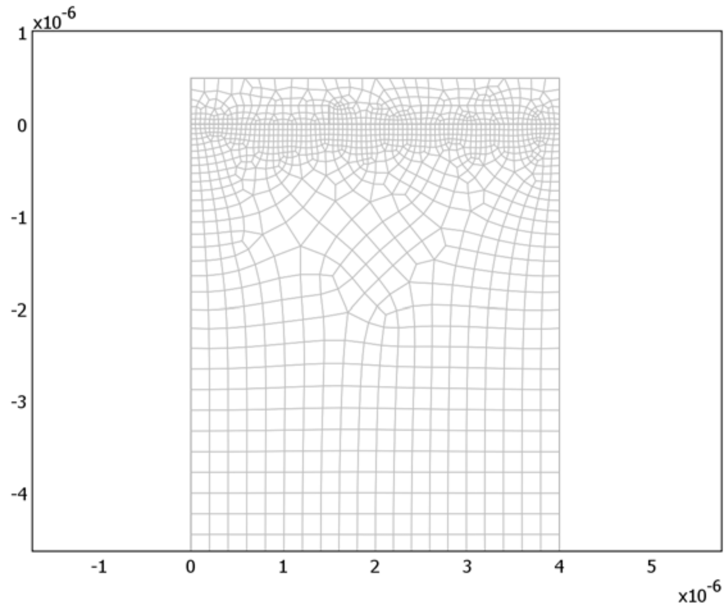


Figure 13-17: The meshed geometry.

COMPUTING THE SOLUTION

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

2 Enter `850e6` in the **Search for eigenfrequencies around** edit field, then click **OK**.

3 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The solver returns 6 eigensolutions with eigenfrequencies in the vicinity of 850 MHz. At 869.8 MHz, two of them are—within the numerical accuracy—the same. These show the shape and the frequency for a SAW with wavelength equal to the width of the geometry.

1 Open the **Plot Parameters** dialog box from the **Postprocessing** menu.

2 On the **General** page, select one of the eigenfrequencies equal to 869.8 MHz.

3 On the **Deform** page, select the **Deformed shape plot** check box. Clear the **Auto** check box and enter `400` in the **Scale factor** edit field.

4 Click **OK** to close the dialog box and see a plot of the total displacement. If you want to, you can repeat the procedure with the other solution to verify that they are the same, only shifted by 90 degrees. One of the solutions will look like Figure 13-7.

5 To evaluate the velocity, choose **Postprocessing>Data Display>Global**.

6 Enter `eigfreq_smppn*4e-6[m]` in the **Expression** edit field.

7 In the **Eigenfrequency** list, select one of the 869.8 MHz eigenfrequencies.

8 Click **OK** to see the value of the velocity in the message log. It evaluates to approximately 3479 m/s.

This concludes the first part of the model. Proceed to find out how the electrodes and the PIB film affect the behavior of the SAW.

Sensor without Gas Exposure

Subdomain Settings

1 Open the **Subdomain Settings** dialog box and select Subdomains 2–4.

2 Select the **Active in this domain** check box.

3 Select **Material model: Decoupled, isotropic**.

4 Select only Subdomain 2.

5 On the **Structural** page, enter `E_PIB` for the **Young's modulus**, `nu_PIB` for the **Poisson's ratio**, and `rho_PIB` for the **Density**.

6 On the **Electrical** page, select the **Enable electrical equation** check box and enter `eps_PIB` for the relative permittivity.

7 Select Subdomains 3 and 4 and click the **Load** button.

8 In the **Materials/Coefficients Library** dialog box, select **Basic Material Properties>Aluminum** and click **OK**.

9 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

1 From the **Physics** menu, choose **Boundary Settings**.

2 Select the **Interior boundaries** check box.

3 Select Boundaries 6-9 and 11-14. On the **Electric BC** page, set the condition to **Electric potential**. Keep the default zero potential.

4 Select Boundaries 3, 5, and 17, and set the condition to **Zero charge/symmetry**.

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

Note: The eigenfrequencies and hence the stopband do not depend on the values of the potentials. In fact, for linear eigenfrequency problems, they are automatically set to zero at the electrodes, regardless of the applied value. You can solve the corresponding driven problem by switching to a frequency response analysis and applying different potentials to the electrodes.

---

6 Choose **Physics>Periodic Conditions>Periodic Boundary Conditions**.
7 On the **Source** page, select Boundary 3. Enter u in the first **Expression** edit field and v in the row below, and V in the third row.
8 On the **Destination** page, select **Constraint name: pconstr1**, check Boundary 17, and enter u in the **Expression** field.
9 Still on the **Destination** page, select **Constraint name: pconstr2**, check Boundary 17, and enter v in the **Expression** field.
10 Finally, select **Constraint name: pconstr3**, check Boundary 17, and enter V in the **Expression** field.
11 Click **OK** to close the dialog box.

You have now established the periodicity in the PIB film.

COMPUTING THE SOLUTION

Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

If you are still using the manual scaling of the deformations from the previous exercise, the plot that appears after solving will look rather distorted. Proceed as follows to find the SAW modes and use more suitable plot parameters:

1 Open the **Plot Parameters** dialog box from the **Postprocessing** menu.
2 On the **General** page, select the 850 MHz eigenfrequency.
3 On the **Deform** page, enter 40 in the **Scale factor** edit field.
4 Click **Apply** to view a plot of the total displacement at anti-resonance.
5 On the **General** page, select the 841 MHz eigenfrequency and click **Apply** to see the deformations at resonance. This plot should look like Figure 13-18.

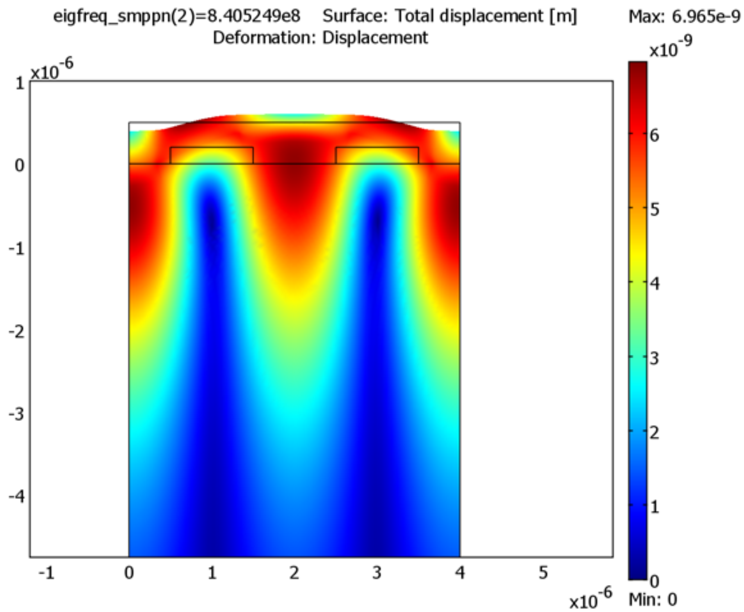


Figure 13-18: Deformations at resonance.

A plot of the electric potential shows a qualitative difference between the two solutions.

6 On the **Surface** page, select **Piezo Plane Strain (smppn)>Electric potential** from the **Predefined quantities** list.

7 Click **Apply** to see the potential distribution at resonance, as shown in Figure 13-15 on page 587. Notice that it is symmetric with respect to each individual electrode.

8 On the **General** page, select the 850 MHz eigenfrequency and click **OK** to see the potential distribution at anti-resonance, as in Figure 13-16 on page 588. This time, it is antisymmetric.

Sensor with Gas Exposure

In the final version of this model, you will expose the sensor to DCM gas. The eigenfrequencies are expected to shift by a very small amount. In order to see the shift, you need to include more digits in the output.

1 Choose **Postprocessing>Data Display>Global**.

2 Enter the expression `eigfreq_smppn` and select the 841 MHz eigenfrequency.

3 Select the **Display result in full precision** check box, then click **OK**.

The message log now shows all computed digits of the eigenfrequency.

Subdomain Settings

1 From the **Physics** menu, select **Subdomain Settings**.

2 Select Subdomain 2. On the **Structural** tab, change the **Density** so that it reads `rho_PIB+rho_DCM_PIB`.

3 Click **OK** to close the **Subdomain Settings** dialog box.

COMPUTING THE SOLUTION

Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

1 Choose **Postprocessing>Data Display>Global**.

2 Make sure that the expression still says `eigfreq_smppn` and select the 841 MHz eigenfrequency.

3 Click **OK**.

The first 6 digits of the eigenfrequency are the same as before. Subtracting the new value from the previous value (which is most easily done by copying and pasting the results from the message log) shows that the eigenfrequency with gas exposure is lower by 227 Hz.

14

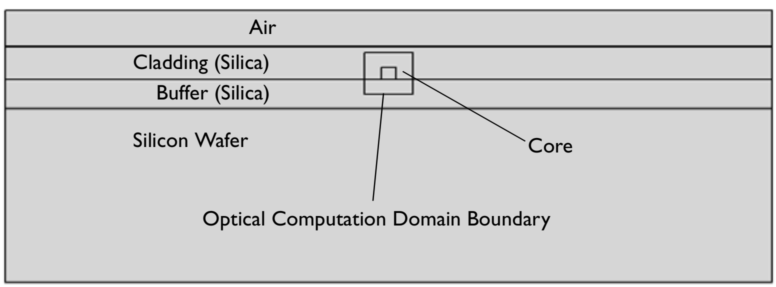
Stress-Optical Effects

This chapter contains models of stress-optical effects.

Stress-Optical Effects in a Silica-on-Silicon Waveguide

Introduction

Planar photonic waveguides in silica (SiO_2) have great potential for use in wavelength routing applications. The major problem with these kinds of waveguides is birefringence. Anisotropic refractive indices result in fundamental mode splitting and pulse broadening. The goal is to minimize birefringence effects by adapting materials and manufacturing processes. One source of birefringence is the use of a silicon (Si) wafer on which the waveguide structure is deposited. After annealing at high temperature (approximately 1000 °C), mismatch in thermal expansivity between the silica and silicon layers results in thermally induced stresses in the structure at the operating temperature (typically room temperature, 20 °C).



The Stress-Optical Effect and Plane Strain

The general linear stress-optical relation can be written, using tensor notation, as

$$\Delta n_{ij} = -B_{ijkl}\sigma_{kl}$$

where $\Delta n_{ij} = n_{ij} - n_0 I_{ij}$, n_{ij} is the refractive index tensor, n_0 is the refractive index for a stress-free material, I_{ij} is the identity tensor, B_{ijkl} is the stress-optical tensor, and σ_{kl} is the stress tensor. Due to symmetry the number of independent parameters in the stress-optical tensor that characterizes this constitutive relation can be reduced. Because n_{ij} and σ_{kl} are both symmetric, $B_{ijkl} = B_{jikl}$ and $B_{ijkl} = B_{ijlk}$. In many cases it is possible to further reduce the number of independent parameters, and this model includes only two independent parameters, B_1 and B_2. The stress-optical relation simplifies to

$$\begin{bmatrix} \Delta n_x \\ \Delta n_y \\ \Delta n_z \end{bmatrix} = -\begin{bmatrix} B_2 & B_1 & B_1 \\ B_1 & B_2 & B_1 \\ B_1 & B_1 & B_2 \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \end{bmatrix}$$

where $n_x = n_{11}$, $n_y = n_{22}$, $n_z = n_{33}$, $\sigma_x = \sigma_{11}$, $\sigma_y = \sigma_{22}$, and $\sigma_z = \sigma_{33}$.

This translates to

$$\begin{aligned} n_x &= n_0 - B_2\sigma_x - B_1(\sigma_y + \sigma_z) \\ n_y &= n_0 - B_2\sigma_y - B_1(\sigma_z + \sigma_x) \\ n_z &= n_0 - B_2\sigma_z - B_1(\sigma_x + \sigma_y) \end{aligned}$$

Using the two parameters B_1 and B_2, the model assumes that the nondiagonal parts of n_{ij} and σ_{kl} are negligible. This means that the shear stress corresponding to $\sigma_{12} = \tau_{xy}$ is neglected. In addition, the shear stresses corresponding to $\sigma_{13} = \tau_{xz}$ and $\sigma_{23} = \tau_{yz}$ are neglected by using the plane strain approximation. The plane strain approximation holds in a situation where the structure is free in the x and y direction but where the z strain is assumed to be zero. Note that this deformation state is not correct if the structure is free also in the z direction. In such a case a modified deformation state equation, which handles the the x and z directions equivalently, is needed.

The first part of this model utilizes the Plane Strain application mode of the Structural Mechanics Module. The resulting birefringent refractive index is computed using expression variables and can be considered a postprocessing step of the plane strain model. The refractive index tensor is used as material data for the second part of the model, the mode analysis.

The model "Stress-Optical Effects with Generalized Plane Strain" on page 615 demonstrates a computation where the structure is free also in the z direction, using a formulation called generalized plane strain.

Perpendicular Hybrid-Mode Waves

For a given frequency ν, or equivalently, free-space wavelength $\lambda_0 = c_0/\nu$, the RF Module's Perpendicular Hybrid-Mode Waves application mode can be used for the mode analysis. In this model the free-space wavelength is 1.55 μm. The simulation is set up with the normalized magnetic field components $\mathbf{H}=(H_x, H_y, H_z)$ as dependent variables, and the effective mode index $n_{\text{eff}} = \beta/k_0$ is obtained from the eigenvalues.

Using this application mode, the wave is assumed to have the form

$$\mathbf{H} = \mathbf{H}(x, y)e^{j(\omega t - \beta z)} = (H_x(x, y), H_y(x, y), H_z(x, y))e^{j(\omega t - \beta z)}$$

The computations show a shift in effective mode index due to the stress-induced change in refractive index. The birefringence causes the otherwise two-fold degenerate fundamental mode to split.

Model Library path: `Structural_Mechanics_Module/Stress-Optical_Effects/stress_optical`

Note: This model requires the RF Module and the Structural Mechanics Module.

Plane Strain Analysis

1 In the **Model Navigator**, select **2D**.

2 Select the **Structural Mechanics Module>Plane Strain>Static analysis** application mode.

3 Click **Multiphysics** then **Add** to add this application mode to the model.

4 Next select the **RF Module>Perpendicular Waves>Hybrid-Mode Waves>Mode analysis** application mode. Click **Add** to add this application mode to the model; then click **OK**.

5 Choose **Physics>Properties** to open the **Application Mode Properties** dialog box and set **Field components** to **In-plane components**. This selects the two-components equation formulation for hybrid-mode waves. The default three-component equation formulation can also be used for this problem. See “Selecting Equation Formulation” on page 146 in the *RF Module User’s Guide* for general information about the different options.

6 For convenience set the property **Specify wave using** to **Free space wavelength**. This makes the wavelength available in the **Application Scalar Variables** dialog box instead of the frequency. Click **OK** to close the **Application Mode Properties** dialog box.

7 Choose **Multiphysics>Model Navigator**. Set the **Ruling application mode** to the **Perpendicular Hybrid-Mode Waves** application mode. This makes this mode specify the interpretation of the parameters to the eigenvalue solver given in the **Solver Parameters** dialog box. Click **OK**.

OPTIONS AND SETTINGS

1 Open the **Constants** dialog box from the **Options** menu and enter the following names,expressions, and descriptions (optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
nSi	3.5	Refractive index, silicon (Si)
nBuf	1.445	Refractive index, silica (SiO_2)
nClad	1.445	Refractive index, silica
deltan	0.0075	Relative index difference: $\Delta = \frac{(n_{\mathrm{core}}^2 - n_{\mathrm{cladding}}^2)}{2n_{\mathrm{core}}^2}$
nCore	nClad/sqrt(1-2*deltan)	Refractive index, core
alphaSi	2.5e-6[1/K]	Thermal expansion coefficient, silicon
alphaSiO2	0.35e-6[1/K]	Thermal expansion coefficient, silica
ESi	110[GPa]	Young's modulus, silicon
ESiO2	78[GPa]	Young's modulus, silica
nuSi	0.19	Poisson's ratio, silicon
nuSiO2	0.42	Poisson's ratio, silica
B1	4.2e-12[m^2/N]	First stress optical coefficient
B2	0.65e-12[m^2/N]	Second stress optical coefficient
T1	20[degC]	Operating temperature
T0	1000[degC]	Reference temperature

Notice that the temperatures are given in degrees Celsius. This works well because this is a linear model. (Nonlinear models need to have temperatures in kelvin to get the correct thermodynamics.)

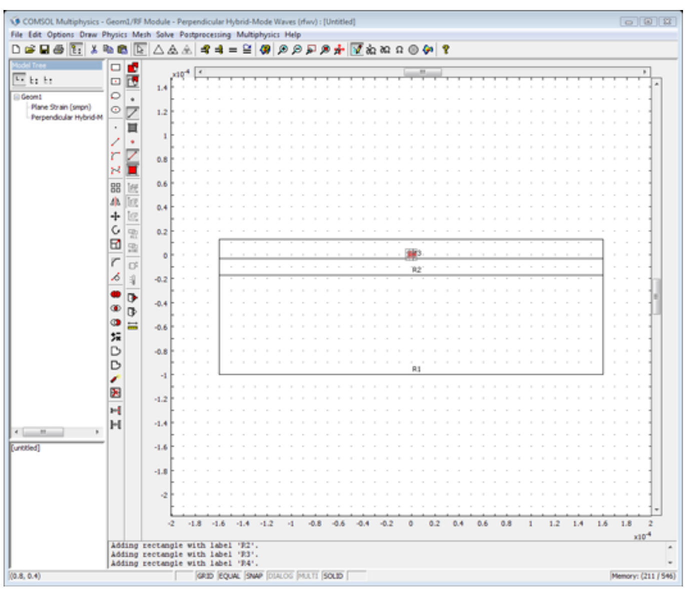
2 Give axis and grid settings according to the following table.

AXIS		GRID	
x min	`-0.2e-3`	x spacing	`1e-5`
x max	`0.2e-3`	Extra x	
y min	`-0.1e-3`	y spacing	`1e-5`
y max	`0.03e-3`	Extra y	

GEOMETRY MODELING

Draw rectangles corresponding to the different regions.

REGION	LOWER LEFT CORNER	UPPER RIGHT CORNER
Silicon Wafer	`(-16e-5,-1e-4)`	`(16e-5,-1.7e-5)`
Buffer	`(-16e-5,-1.7e-5)`	`(16e-5,-3e-6)`
Cladding	`(-16e-5,-3e-6)`	`(16e-5,1.3e-5)`
Core	`(-3e-6,-3e-6)`	`(3e-6,3e-6)`



PHYSICS SETTINGS

Point Settings and Boundary Conditions

All regions have free boundaries, which also is the default boundary condition. However, these conditions will not suffice in creating a unique solution because the computational domain is allowed to move and rotate freely; the problem is ill-posed. The problem becomes well-posed by adding constraints at points to keep the domain fixed.

1 Select the **Plane Strain** application mode from the **Multiphysics** menu.
2 Open the **Point Settings** dialog box, and check the constraints R_x and R_y at Point 1, the lower left corner, for both the x and y direction. This will keep the domain from moving (translational move).
3 Check the constraint R_y at Point 9, the lower right corner, for the y direction only, keeping the domain from rotating but allowing it to slide in the x direction.

Subdomain Settings

The air domain need not be part of the plane strain model.

1 Specify the subdomain settings for the Plane Strain application mode according to the following table:

	SUBDOMAIN 1				SUBDOMAINS 2–4			
Page	Material		Load		Material		Load	
	E	`ESi`			E	`ESiO2`		
	ν	`nuSi`	Temp	`T1`	ν	`nuSiO2`	Temp	`T1`
	α	`alphaSi`	Tempref	`T0`	α	`alphaSiO2`	Tempref	`T0`

Before entering the temperatures on the **Load** page, you need to check **Include thermal expansion**. It is not necessary to specify the density ρ, because it does not enter the equation for static problems.

EXPRESSION VARIABLES

In the **Subdomain Expressions** dialog box in the **Options** menu, define the following variables; when done, click **OK**.

VARIABLE NAME	SUBDOMAIN	EXPRESSION
N	1 2 3 4	nSi nBuf nClad nCore
Nx	1 2,3,4	N N-B1*sx_smpn-B2*(sy_smpn+sz_smpn)
Ny	1 2,3,4	N N-B1*sy_smpn-B2*(sx_smpn+sz_smpn)
Nz	1 2,3,4	N N-B1*sz_smpn-B2*(sx_smpn+sy_smpn)

MESH GENERATION

Initialize a mesh and refine it twice.

COMPUTING THE SOLUTION

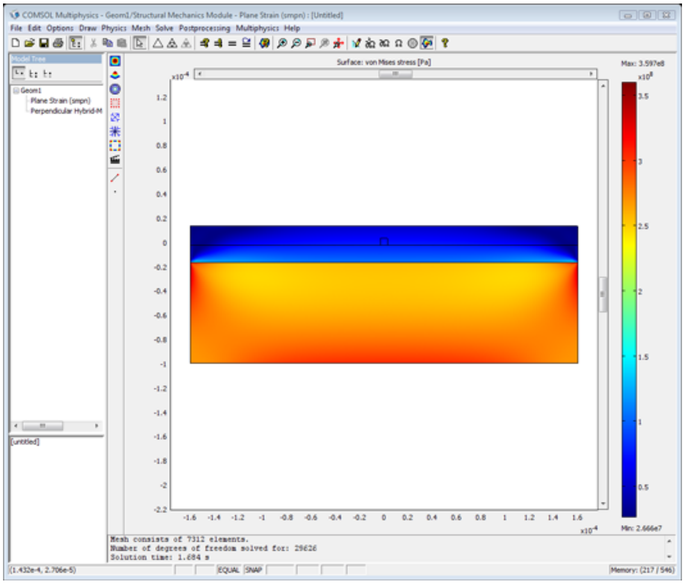
1 In the **Solver Parameters** dialog box select the **Stationary** solver.

2 On the **Solve for** tab in the **Solver Manager** dialog box, select only the **Plane Strain** application mode. This will make sure that only the Plane Strain application mode equations are solved in the first run.

3 Click the **Solve** button.

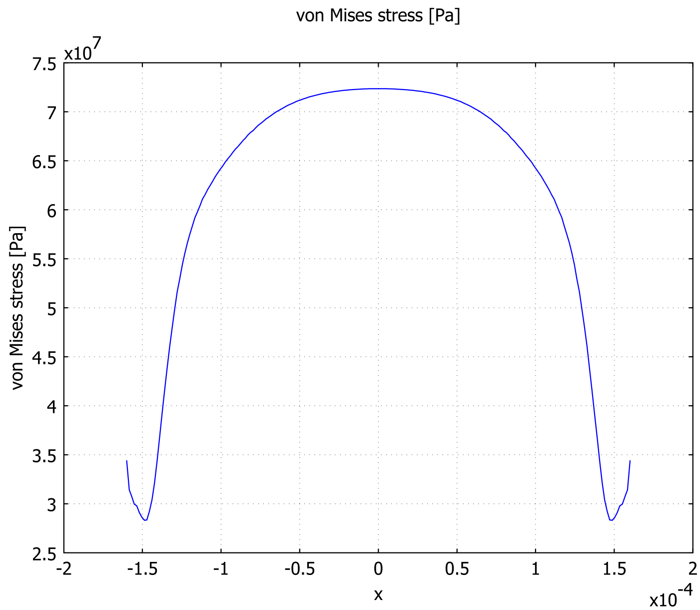
POSTPROCESSING AND VISUALIZATION

The default plot shows the von Mises effective stress as a colored surface plot.



1 To view the von Mises stress along a horizontal line through the entire structure, use a cross-section plot. In the **Cross-Section Plot Parameters** dialog box, select **von**

Mises stress as **y-axis data** on the **Line/Extrusion** tab. As **Cross-section line data** set **x0** to `-16e-5`, **y0** to `0`, **x1** to `16e-5`, and **y1** to `0`. Use `x` as **x-axis data**.

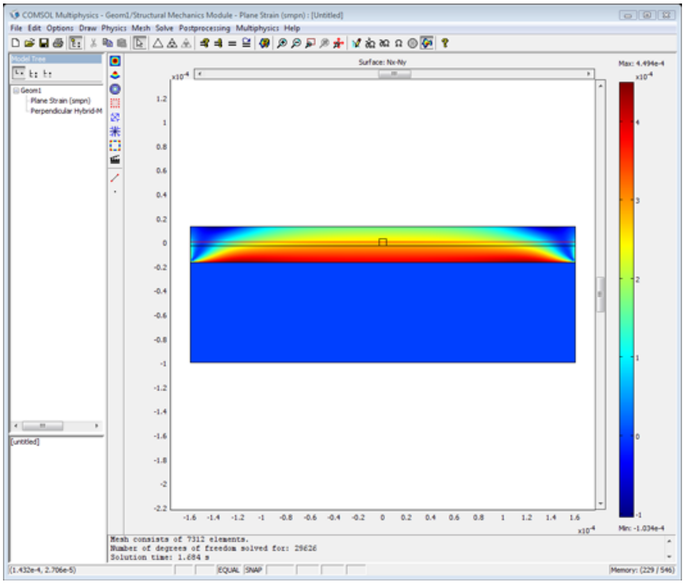


Notice that the effective stress is varying slowly on the horizontal cut. This means that the significant influence on the stress induced changes in refractive index will come from the stress variations in the vertical direction. This is expected since the extension of the domains in the *x* direction is chosen to minimize effects of the edges.

Next plot the birefringence:

2 In the **Plot Parameters** dialog box, click the **Surface** tab.

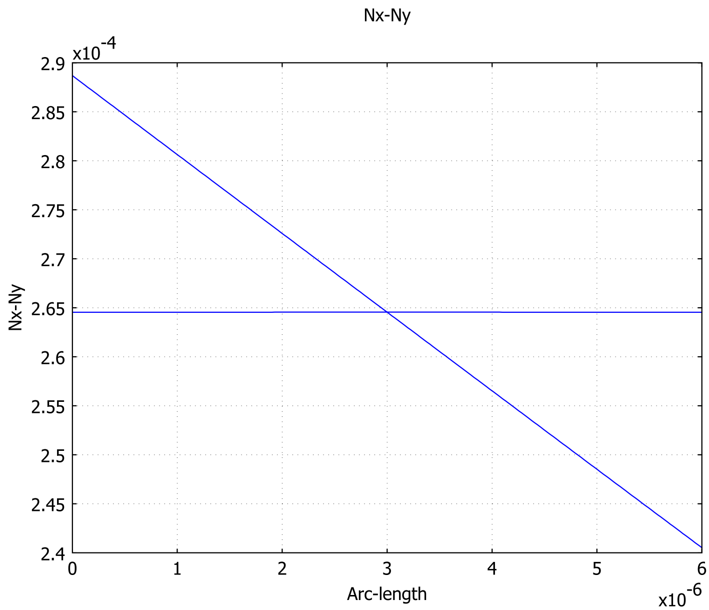
3 To view the stress-induced birefringence $n_x - n_y$, type `Nx-Ny` in the **Expression** field on the **Surface Data** page.



4 Now create a cross-section plot of the birefringence $n_x - n_y$ on the vertical line from $(0, -3{\cdot}10^{-6})$ to $(0, 3{\cdot}10^{-6})$. Do this by entering `Nx-Ny` as **y-axis data**, and setting **x0** to `0`, **y0** to `-3e-6`, **x1** to `0`, and **y1** to `3e-6` in the **Cross-Section Plot Parameters** dialog box. Set the **x-axis data** to **Arc-length**. The birefringence varies linearly from about $2.9{\cdot}10^{-4}$ at the bottom of the core to $2.4{\cdot}10^{-4}$ at the top. Do not close the figure.

5 Now create another cross-section plot of the birefringence on the horizontal line from $(-3{\cdot}10^{-6}, 0)$ to $(3{\cdot}10^{-6}, 0)$. On the **General** tab select the **Keep current plot**

check box to make the plot in the same window as the previous one. On the **Line/ Extrusion** page, set **x0** to `-3e-6`, **y0** to `0`, **x1** to `3e-6`, and **y1** to `0`.



The birefringence is constant on the horizontal line, thus the influence of the edges is indeed reduced to a minimum.

Optical Mode Analysis

Select the **Perpendicular Hybrid-Mode Waves** application mode from the **Multiphysics** menu.

GEOMETRY MODELING

The computational domain can be reduced significantly for the optical mode analysis, because the energy of the fundamental modes is concentrated in the core region, and the energy density decays rapidly in the cladding and buffer regions.

Draw a rectangle with corners at (–1e-5, –1e-5) and (1e-5, 1e-5).

The rectangular region containing the active domains can be enlarged later on for validating the results. The rectangular region should be chosen large enough so that the computed propagation constants do not change significantly if the region is enlarged.

PHYSICS SETTINGS

Scalar Variables

1 From the **Physics** menu, choose **Scalar Variables**.

2 In the **Application Scalar Variables** dialog box, set the free-space wavelength to `1.55e-6`; when done, click **OK**.

Subdomain Settings

1 From the **Physics** menu, choose **Subdomain Settings**.

2 Deactivate the Perpendicular Hybrid-Mode Waves application mode in all subdomains except 4, 5, and 6. This makes sure the problem is only solved in the newly drawn rectangle.

3 For all active domains, select **n (anisotropic)** and set `Nx`, `Ny`, and `Nz`, respectively, as anisotropic refractive indices by changing the entries on the diagonal in the refractive index matrix.

4 When done, click **OK**.

Boundary Conditions

The only available perfect magnetic conductor boundary condition will suffice. Both the **E** and the **H** fields are assumed to be of negligible size at the boundaries, hence any one of them can be set equal to zero at the boundary.

MESH GENERATION

1 On the **Subdomain** tab in the **Free Mesh Parameters** dialog box, set the **Maximum element size** to `2e-6` for Subdomains 4 and 5, and to `1e-6` for Subdomain 6. This will make the mesh dense in the activated regions and coarse in the deactivated regions. The mesh in the deactivated regions only affects the interpolated values of the expression variables and not the mode computation results.

2 Initialize the mesh.

COMPUTING THE SOLUTION

1 In the **Solver Parameters** dialog box, select the **Eigenvalue** solver.

2 Enter `1.46` in the **Search for effective mode indices around** edit field. The default **Desired number of effective mode indices** is set to `6`, which is fine. These settings will make the eigenmode solver search for the 6 eigenmodes with effective mode indices closest to the value `1.46`. This value is an estimate of the effective mode index for the fundamental mode. For propagating modes it must hold that

$$n_{\mathrm{eff}} < n_{\mathrm{core}} = 1.456$$

3 In the **Solver Manager** dialog box, select only the **Perpendicular Hybrid-Mode Waves** application mode on the **Solve For** tab. This ensures that the Plane Strain application mode is not part of the eigenmode computation.

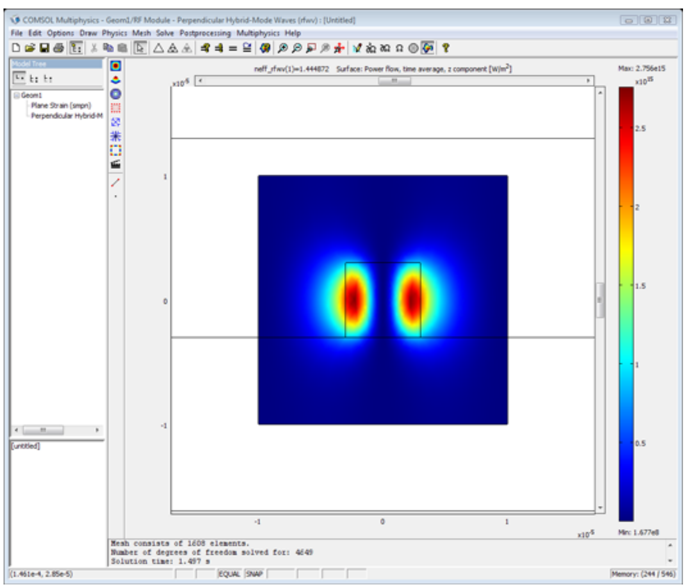
4 On the **Initial Value** tab, set the initial value to **Current solution**. This will take the plane strain solution and use it to evaluate the refractive indices for the mode analysis.

5 Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

1 On the **Surface** tab in the **Plot Parameters** dialog box, select **Perpendicular Hybrid-Mode Waves (rfwv)>Power flow, time average, z component** from the **Predefined quantities** list on the **Surface Data** tab. Then click **Apply**. This creates a visualization of the power flow, also called optical intensity or the Poynting vector, in the *z* direction (out-of-plane direction).

2 The default eigenmode is the one with the highest effective mode index corresponding to one of the fundamental modes.



Convergence Analysis

To check the accuracy of the eigenvalues you need to examine the sensitivity of these values to changes in the mesh density.

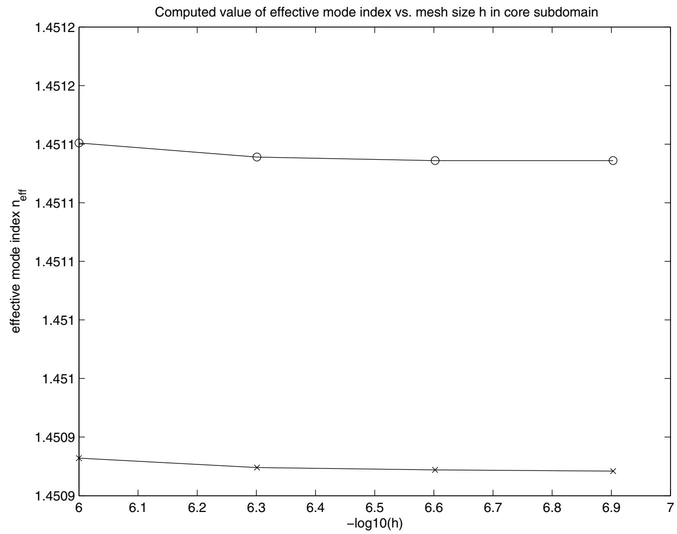
1 In the **Free Mesh Parameters** dialog box, lower the value of the maximum element size for the core subdomain, number 6, to `5e-7`, remesh, and solve again.

2 Repeat this for the maximum element sizes `2.5e-7` and `1.25e-7`.

The solver finds the following effective mode:

ELEMENT SIZE, CORE SUBDOMAIN EFFECTIVE MODE INDEX	1E-6	5E-7	2.5E-7	1.25E-7
n_{eff1}	1.451149	1.451139	1.451136	1.451136
n_{eff2}	1.450883	1.450875	1.450872	1.450871
n_{eff3}	1.445409	1.445407	1.445405	1.445405
n_{eff4}	1.445144	1.445141	1.445139	1.445139
n_{eff5}	1.445139	1.445133	1.445132	1.445131
n_{eff6}	1.444873	1.444867	1.444865	1.444865

It is evident from the table that the fundamental modes, corresponding to the two lowest eigenmodes with the highest effective mode indices, are nearly degenerate. It is also clear that the split of the degenerate fundamental modes that is expected due to the stress-induced anisotropic refractive index has been properly resolved with these mesh sizes. The convergence of the two fundamental modes is shown in the figure below.



Now, run the model without the stress-induced birefringence.

1. Change the refractive index of Subdomains 4, 5, and 6 to an isotropic index by selecting the **n (isotropic)** radio button in the **Subdomain Settings** dialog box, and typing `N` as refractive index.
2. Solve the problem for a maximum element size of `2.5e-7` for Subdomain 6.

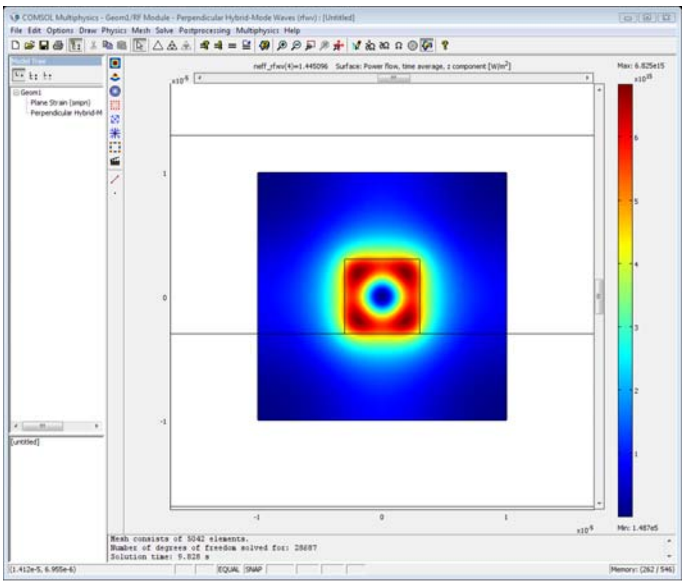
Compare these ideal values of the propagation constants with the stress-optical case.

ELEMENT SIZE, CORE SUBDOMAIN: EFFECTIVE MODE INDEX	2.5E-7 (STRESS)	2.5E-7 (NO STRESS)	DIFFERENCE *1e-4
n_{eff1}	1.451136	1.450821	3.15
n_{eff2}	1.450872	1.45082	.51
n_{eff3}	1.445405	1.445093	57.79
n_{eff4}	1.445139	1.445083	.56
n_{eff5}	1.445132	1.444832	3.00
n_{eff6}	1.444865	1.444799	.66

The difference is significant, which shows that the shift in the effective mode indices due to the stress-optical effect is indeed resolved.

Visual inspection of the 4 higher eigenmodes indicates that these are probably degenerate in the ideal case of no thermally induced stresses. Moreover, the higher eigenmodes have a larger portion of energy leaking into the cladding and buffer, and are thus more affected than the fundamental modes of the distance to the air and silicon layers. Because of this leakage, the boundary condition will affect the higher eigenmodes more than the fundamental mode. Thus a refined analysis of the higher modes would make it necessary to enlarge the computation domain.

The figure below shows the *z* component of the Poynting vector for the fourth eigenmode.



MODELING ERRORS AND SENSITIVITY ANALYSIS

Although the fundamental modes have converged to 5 decimal places, the known modeling errors makes the exactness of the numbers uncertain. One major modeling error is due to the fact that the model contains a plane strain assumption in a case where the real-world model does not necessarily conform to this deformation state. This modeling error is reduced in the refined model "Stress-Optical Effects with Generalized Plane Strain" on page 615. Moreover, the material properties are only known to a few decimal places and the computed magnitudes of the effective mode indices will correspondingly be uncertain. A standard way of examining the effects of uncertainty in material parameters is to perform a sensitivity analysis. That is, you perturb one of the material parameters slightly and then examine the resulting perturbation in the computed parameter. Another source of uncertainty is whether the stress from the thermal expansion is large with respect to the other sources of stress in the material originating from the manufacturing process.

Stress-Optical Effects with Generalized Plane Strain

Introduction

The assumptions made for plane strain in the previous analysis of the waveguide structure, in the model "Stress-Optical Effects in a Silica-on-Silicon Waveguide" on page 598, do not hold in a situation where the silicon-silica laminate is free to expand in the z direction. Instead it is necessary to use a *generalized plane strain* model that allows for free expansion in the z direction. The boundary conditions in the xy-plane already allow the structure to expand freely in all directions in the plane. When the different materials in a laminate expand with different expansion coefficients, the laminate bends. In this model, the silica-silicon laminate bends in both the x and z directions. The Plane Strain application mode does not cover the bending in the z direction, so you must make modifications to the plane strain equations at the equation-system level.

Note: This model requires the RF Module and the Structural Mechanics Module.

Generalized Plane Strain

One possible extension of the plane strain formulation is to assume that the strain in the z direction has the form

$$\varepsilon_z = e_0 + e_1 x + e_2 y$$

That is, the strain is linearly varying throughout the cross section. This approximation is expected to be good when the bending curvature is small with respect to the extents of the structure in the xy-plane and corresponds to a small rotation that is representative of each cross section of the structure along the z axis. (A more general model would include second-degree terms in x and y.)

One way of validating the result of this simulation is to plot the ε_x and ε_z strains on a cross section on the symmetry line at $x = 0$. Because you can expect an identical bending in the x and z directions, these strains should be identical and vary linearly.

EXTENSION OF THE PLANE STRAIN EQUATIONS

The objective is to extend the current Plane Strain application mode with additional PDE terms that represent the generalized plane strain deformation state. In COMSOL Multiphysics, the coefficients e_0, e_1, and e_2 in the expression for the ε_z strain is modeled with coupling variables associated with a point geometry object (zero dimensional object). This is a convenient way to include degrees of freedom that are constant throughout a subdomain. In order to expand the equations for the Plane Strain application mode, the new ε_z strain contributions need to be added as weak terms to the ordinary plane strain equations. The reason for this is that for COMSOL Multiphysics to be able to solve the equations as a linear problem, the assembler requires that all contributions from coupling variables are entered as weak terms.

Start from the 3D stress-strain relation for linear isotropic conditions including thermal effects,

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{bmatrix} = D\left(\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix} - \alpha\begin{bmatrix} T - T_{\mathrm{ref}} \\ T - T_{\mathrm{ref}} \\ T - T_{\mathrm{ref}} \\ 0 \\ 0 \\ 0 \end{bmatrix}\right)$$

where σ_x, σ_y, σ_z, τ_{xy}, τ_{yz}, and τ_{xz} are the components of the stress tensor, ε_x, ε_y, ε_z, γ_{xy}, γ_{yz}, and γ_{xz} are the components of the strain tensor, α is the thermal expansion coefficient, T is the operating temperature, and T_{ref} is the reference temperature corresponding to the manufacturing temperature. The 6-by-6 matrix $\boldsymbol{D}$ (tensor) has entries that contain expressions in Young's modulus, E, and Poisson's ratio, ν. The matrix $\boldsymbol{D}$ is presented in the *Structural Mechanics Module User's Guide*.

For generalized plane strain (and ordinary plane strain), assume that $\gamma_{yz} = \gamma_{xz} = 0$ and two of the equations vanish

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \end{bmatrix} = D\left(\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \end{bmatrix} - \alpha\begin{bmatrix} T - T_{\mathrm{ref}} \\ T - T_{\mathrm{ref}} \\ T - T_{\mathrm{ref}} \\ 0 \end{bmatrix}\right)$$

The matrix $\boldsymbol{D}$ shrinks accordingly.

The PDE solved is Navier's equation

$$-\nabla \cdot \sigma = \mathbf{K}$$

where

$$\sigma = \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} = c\nabla\mathbf{u} - \gamma$$

in the case of generalized plane strain and $\mathbf{u} = (u, v, w)$ are the unknown displacement variables solved for.

The coefficient c above is a tensor containing the elements of D in a pattern that is listed in the *Structural Mechanics Module User's Guide* and

$$\gamma = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} (1+\nu)\alpha(T-T_{\text{ref}}) & 0 & 0 \\ 0 & (1+\nu)\alpha(T-T_{\text{ref}}) & 0 \\ 0 & 0 & (1+\nu)\alpha(T-T_{\text{ref}}) \end{bmatrix}$$

contains the contributions from the thermal strains.

In the case of ordinary plane strain the upper left 2-by-2 submatrix of σ, γ, and D is used.

It is assumed here that the variables T and T_{ref} are constants. For information on how the temperature variables enter Navier's equation as unknowns in a heat transfer problem, see the *Structural Mechanics Module User's Guide*.

To expand the plane strain equation to a generalized plane strain formulation one needs to consider the weak form of Navier's equation. The weak form of a system of equations is a scalar equation involving integrals. See "Weak Form Modeling" on page 293 in the *COMSOL Multiphysics Modeling Guide* for more information on the appearance of the weak form for systems of equations.

To get the weak form of Navier's equation, multiply by a test function $\mathbf{v} = [u_{\text{test}}, v_{\text{test}}, w_{\text{test}}]$ and integrate

$$\int_\Omega \mathbf{v}(-\nabla \cdot \sigma)d\Omega = \int_\Omega \mathbf{v} \cdot \mathbf{K}d\Omega$$

Integration by parts gives

$$0 = \int_{\partial\Omega} \mathbf{v}\sigma\mathbf{n}d\Gamma - \int_{\Omega} \nabla\mathbf{v} \cdot \sigma d\Omega + \int_{\Omega} \mathbf{v} \cdot \mathbf{K} d\Omega$$

or in more detail, with $\mathbf{K} = \mathbf{0}$

$$0 = \int_{\partial\Omega} \begin{bmatrix} u_{\text{test}} & v_{\text{test}} & w_{\text{test}} \end{bmatrix} \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} \begin{bmatrix} n_x \\ n_y \\ n_z \end{bmatrix} d\Gamma$$

$$- \int_{\Omega} \left(\begin{bmatrix} u_{x,\,\text{test}} & u_{y,\,\text{test}} & u_{z,\,\text{test}} \\ v_{x,\,\text{test}} & v_{y,\,\text{test}} & v_{z,\,\text{test}} \\ w_{x,\,\text{test}} & w_{y,\,\text{test}} & w_{z,\,\text{test}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} \right) d\Omega$$

where $u_{y,\text{test}} = \partial u_{\text{test}} / \partial y$ and similarly for the other test function derivatives. The dot product in the domain integral is interpreted as a dot product operating on the rows of the matrices so that

$$\int_{\Omega} \left(\begin{bmatrix} u_{x,\,\text{test}} & u_{y,\,\text{test}} & u_{z,\,\text{test}} \\ v_{x,\,\text{test}} & v_{y,\,\text{test}} & v_{z,\,\text{test}} \\ w_{x,\,\text{test}} & w_{y,\,\text{test}} & w_{z,\,\text{test}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} \right) d\Omega$$

$$= \int_{\Omega} \Big(\begin{bmatrix} u_{x,\,\text{test}} & u_{y,\,\text{test}} & u_{z,\,\text{test}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \end{bmatrix} +$$

$$\begin{bmatrix} v_{x,\,\text{test}} & v_{y,\,\text{test}} & v_{z,\,\text{test}} \end{bmatrix} \cdot \begin{bmatrix} \tau_{yx} & \sigma_y & 0 \end{bmatrix} + \begin{bmatrix} w_{x,\,\text{test}} & w_{y,\,\text{test}} & w_{z,\,\text{test}} \end{bmatrix} \cdot \begin{bmatrix} 0 & 0 & \sigma_z \end{bmatrix} \Big) d\Omega$$

where

$$\sigma_x = \frac{E}{(1+\nu)(1-2\nu)}((1-\nu)\varepsilon_x + \nu\varepsilon_y + \nu\varepsilon_z - (1+\nu)\alpha(T - T_{\mathrm{ref}}))$$
$$\sigma_y = \frac{E}{(1+\nu)(1-2\nu)}(\nu\varepsilon_x + (1-\nu)\varepsilon_y + \nu\varepsilon_z - (1+\nu)\alpha(T - T_{\mathrm{ref}}))$$
$$\sigma_z = \frac{E}{(1+\nu)(1-2\nu)}(\nu\varepsilon_x + \nu\varepsilon_y + (1-\nu)\varepsilon_z - (1+\nu)\alpha(T - T_{\mathrm{ref}}))$$
$$\tau_{xy} = \frac{E}{(1+\nu)(1-2\nu)}\left(\frac{1-2\nu}{2}\gamma_{xy}\right)$$

and

$$\varepsilon_x = \frac{\partial u}{\partial x}$$
$$\varepsilon_y = \frac{\partial v}{\partial y}$$
$$\varepsilon_z = \frac{\partial w}{\partial z}$$
$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}$$

Note that the displacement variable w in the z direction is not available in the Plane Strain application mode. However, it is not needed since the generalized formulation assumes a certain fix shape of the deformation in the z direction.

The same relations hold for the test functions,

$$\varepsilon_{x,\,\mathrm{test}} = u_{x,\,\mathrm{test}}$$
$$\varepsilon_{y,\,\mathrm{test}} = v_{y,\,\mathrm{test}}$$
$$\varepsilon_{z,\,\mathrm{test}} = w_{z,\,\mathrm{test}}$$
$$\gamma_{xy,\,\mathrm{test}} = u_{y,\,\mathrm{test}} + v_{x,\,\mathrm{test}}$$

In COMSOL Multiphysics, the test functions u_{test} and v_{test} are denoted `u_test` and `v_test`, respectively. Their derivatives $u_{x,\mathrm{test}}$, $u_{y,\mathrm{test}}$, $v_{x,\mathrm{test}}$, and $v_{y,\mathrm{test}}$ are denoted `ux_test`, `uy_test`, `vx_test`, and `vy_test`, respectively. Note that the z components of the test functions are not available in the Plane Strain application mode.

Using this notation, the weak form of Navier's equation becomes (with $\mathbf{K} = \mathbf{0}$)

$$0 = \int_{\partial\Omega} \mathbf{v}\sigma\mathbf{n}\,d\Gamma - \int_{\Omega} (\varepsilon_{x,\,\mathrm{test}}\sigma_x + \varepsilon_{y,\,\mathrm{test}}\sigma_y + \varepsilon_{z,\,\mathrm{test}}\sigma_z + \gamma_{xy,\,\mathrm{test}}\tau_{xy})\,d\Omega$$

This expression for the weak equation shows which terms you must add as weak terms to generalize the plane strain formulation.

To see the difference compared to the ordinary plane strain formulation, the equation for σ_z can be separated out according to

$$\begin{bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{bmatrix} = D\left(\begin{bmatrix}\varepsilon_x\\ \varepsilon_y\\ \gamma_{xy}\end{bmatrix} - \begin{bmatrix}(1+\nu)\alpha(T-T_{\mathrm{ref}})\\ (1+\nu)\alpha(T-T_{\mathrm{ref}})\\ 0\end{bmatrix} + \begin{bmatrix}\nu\varepsilon_z\\ \nu\varepsilon_z\\ 0\end{bmatrix}\right)$$

$$\sigma_z = \frac{E}{(1+\nu)(1-2\nu)}(\nu\varepsilon_x + \nu\varepsilon_y + (1-\nu)\varepsilon_z - (1+\nu)\alpha(T-T_{\mathrm{ref}}))$$

where D has been reduced to

$$D = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix}1-\nu & \nu & 0\\ \nu & 1-\nu & 0\\ 0 & 0 & \frac{1-2\nu}{2}\end{bmatrix}$$

The ordinary plane strain equation is obtained if $\varepsilon_z = 0$.

To expand the Plane Strain application mode, you must add weak contributions from ε_z need to the stress components σ_x and σ_y according to

$$-\frac{E}{(1+\nu)(1-2\nu)}(\varepsilon_{x,\,\mathrm{test}}\nu\varepsilon_z + \varepsilon_{y,\,\mathrm{test}}\nu\varepsilon_z)$$

In addition, σ_z needs to be included in the weak equation from being merely a postprocessing quantity in the original Plane Strain application mode,

$$-\varepsilon_{z,\,\mathrm{test}}\sigma_z$$

where

$$\varepsilon_{z,\,\mathrm{test}} = e_{0,\,\mathrm{test}} + e_{1,\,\mathrm{test}}x + e_{2,\,\mathrm{test}}y$$

and σ_z was presented earlier.

In COMSOL Multiphysics you model the coefficients e_0, e_1, and e_2 in the expression for the ε_z strain with coupling variables as weak contributions to the ordinary plane strain equations. These coefficients represent three additional unknown variables that prevail throughout the modeled domain. In practice, these variables need to be associated with an arbitrary point of the geometry. Using coupling variables in this way specifies a map from this point onto the entire domain on which the generalized plane strain equations hold. COMSOL Multiphysics creates the test functions corresponding to e_0, e_1, and e_2 automatically.

Model Library path: `Structural_Mechanics_Module/Stress-Optical_Effects/stress_optical_generalized`

Plane Strain Analysis

Many of the modeling steps are similar to the model “Stress-Optical Effects in a Silica-on-Silicon Waveguide” on page 598, but some steps are made in a different order, and there are obviously differences when generalizing the plane strain problem.

MULTIPHYSICS SETTINGS

1 In the **Model Navigator**, select **2D**.

2 Select the **Structural Mechanics Module>Plane Strain>Static analysis** application mode.

3 Click **Multiphysics** then **Add** to add this application mode to the model.

4 Next select the **RF Module>Perpendicular Waves>Hybrid-Mode Waves> Mode analysis** application mode. Add this mode to the model.

5 Open the **Application Mode Properties** dialog box from the **Physics** menu (**Properties**) and select **In-plane components** from the **Field components** list. This selects the two-components equation formulation for hybrid-mode waves. It is also possible to use the default three-component equation formulation for this problem. See “Selecting Equation Formulation” on page 146 in the *RF Module User’s Guide* for general information about the different options.

6 For convenience select **Free space wavelength** from the **Specify wave using** list. This makes the wavelength available in the **Application Scalar Variables** dialog box instead of the frequency.

The unknown parameters e_0, e_1, and e_2 should be constant within the whole modeling domain. You can achieve this by adding a new geometry with only a single point to the model and define e_0, e_1, and e_2 as the dependent variables at this point. Later on you define coupling variables that map these dependent variables from the point to the whole model.

1 From the **Multiphysics** menu, select the **Model Navigator**. Click the **Add Geometry** button to add a 2D geometry (2D is the default space dimension).

2 Select the **COMSOL Multiphysics>PDE Modes>Weak Form,Point** application mode from the application mode tree.

3 Type `e0 e1 e2` in the **Dependent variables** text field.

4 Click **Add** to add the Weak Form, Point application mode to the second geometry. This application mode contains the three degrees of freedom for the parameters e_0, e_1, and e_2 that describe the z strain.

5 Set the **Ruling application mode** to the **Perpendicular Hybrid-Mode Waves** application mode. This makes this application mode specify the interpretation of the parameters to the eigenvalue solver given in the **Solver Parameters** dialog box.

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

The Weak Form, Point application mode becomes the active application mode when you close the Model Navigator.

GEOMETRY MODELING

The three degrees of freedom e_0, e_1, and e_2 must be associated with a point. The location of this point is insignificant. This means that the added geometry could equally well have been a 1D or a 3D geometry. Using the programming language, there is also an option of using a 0D geometry.

Add a point at any position, say at the origin (0, 0).

OPTIONS AND SETTINGS

In the **Constants** dialog box enter the following names, expressions, and descriptions (optional); when done, click **OK**. They are identical to the constants defined on page 601.

NAME	EXPRESSION	DESCRIPTION
`nSi`	`3.5`	Refractive index, silicon (Si)
`nBuf`	`1.445`	Refractive index, silica (SiO_2)
`nClad`	`1.445`	Refractive index, silica
`deltan`	`0.0075`	Relative index difference: $\Delta = \frac{(n_{\mathrm{core}}^2 - n_{\mathrm{cladding}}^2)}{2n_{\mathrm{core}}^2}$
`nCore`	`nClad/sqrt(1-2*deltan)`	Refractive index, core

NAME	EXPRESSION	DESCRIPTION
alphaSi	2.5e-6[1/K]	Thermal expansion coefficient, silicon
alphaSiO2	0.35e-6[1/K]	Thermal expansion coefficient, silica
ESi	110[GPa]	Young's modulus, silicon
ESiO2	78[GPa]	Young's modulus, silica
nuSi	0.19	Poisson's ratio, silicon
nuSiO2	0.42	Poisson's ratio, silica
B1	4.2e-12[m^2/N]	First stress optical coefficient
B2	0.65e-12[m^2/N]	Second stress optical coefficient
T1	20[degC]	Operating temperature
T0	1000[degC]	Reference temperature

GEOMETRY MODELING

1 Switch to **Geom1** by clicking its tab.

2 Draw rectangles corresponding to the different regions.

REGION	LOWER LEFT CORNER	UPPER RIGHT CORNER
Silicon Wafer	(-16e-5,-1e-4)	(16e-5,-1.7e-5)
Buffer	(-16e-5,-1.7e-5)	(16e-5,-3e-6)
Cladding	(-16e-5,-3e-6)	(16e-5,1.3e-5)
Core	(-3e-6,-3e-6)	(3e-6,3e-6)

PHYSICS SETTINGS

Point Settings and Boundary Conditions

All regions have free boundaries, which also is the default boundary condition. However, these conditions do not suffice in creating a unique solution because the computational domain can move and rotate freely; the problem is ill-posed. The problem becomes well-posed by adding constraints at points to keep the domain fixed.

1 Select the **Plane Strain** application mode from the **Multiphysics** menu.

2 Open the **Point Settings** dialog box, and select the check boxes for the constraints R_x and R_y at point 1, the lower left corner, for both the x and y directions. Doing so keeps the domain from moving (translational move).

3 Select the check box for the constraint R_y at point 9, the lower-right corner, for the y direction only, keeping the domain from rotating but allowing it to slide in the x direction. When done, click **OK**.

Subdomain Settings

The air domain need not be part of the plane strain model.

1 Specify the subdomain settings for the Plane Strain application mode according to the following table.

	SUBDOMAIN 1				**SUBDOMAINS 2–4**			
Page	Material		Loads		Material		Loads	
	E	`ESi`			E	`ESiO2`		
	ν	`nuSi`	Temp	`T1`	ν	`nuSiO2`	Temp	`T1`
	α	`alphaSi`	Tempref	`T0`	α	`alphaSiO2`	Tempref	`T0`

Before entering the temperatures on the **Load** page, you must select the **Include thermal expansion** check box. It is not necessary to specify the density ρ, because it does not enter the equation for static problems.

EXPRESSION VARIABLES

In the **Subdomain Expressions** dialog box in the **Options** menu, define the following variables.

VARIABLE NAME	SUBDOMAIN	EXPRESSION
`E`	1 2,3,4	`ESi` `ESiO2`
`nu`	1 2,3,4	`nuSi` `nuSiO2`
`alpha`	1 2,3,4	`alphaSi` `alphaSiO2`
`ex`	1,2,3,4	`ux`
`ey`	1,2,3,4	`vy`
`ez`	1,2,3,4	`e00+e11*x+e22*y`
`sx`	1,2,3,4	`sx_smpn+E/((1+nu)*(1-2*nu))*nu*ez`
`sy`	1,2,3,4	`sy_smpn+E/((1+nu)*(1-2*nu))*nu*ez`
`sz`	1,2,3,4	`E/((1+nu)*(1-2*nu))*(nu*ex+nu*ey+(1-nu)*ez-(1+nu)*alpha*(T1-T0))`

`N`	1
	2
	3
	4
		`nSi`
		`nBuf`
		`nClad`
		`nCore`

`Nx`	1 2,3,4	`N` `N-B1*sx-B2*(sy+sz)`
`Ny`	1 2,3,4	`N` `N-B1*sy-B2*(sx+sz)`
`Nz`	1 2,3,4	`N` `N-B1*sz-B2*(sx+sy)`

The variables `ex`, `ey`, and `ez` are the strains ε_x, ε_y, and ε_z, and the variables `sx`, `sy`, and `sz` are the stresses σ_x, σ_y, and σ_z. The definitions of the refractive index variables `Nx`, `Ny`, and `Nz` differ slightly compared to the previous model. In this case they are expressed in the generalized stress variables `sx`, `sy`, and `sz`. Because of the use of integration coupling variables (`e00`, `e11`, and `e22`), COMSOL Multiphysics cannot determine the unit for other variables that directly or indirectly depend on these integration coupling variables. This causes the warnings for inconsistent units here and in the specification of the refractive index. You can disregard these warnings.

COUPLING VARIABLES

The unknown parameters e_0, e_1, and e_2 should be constant within the whole modeling domain. You can achieve this by defining coupling variables that map these dependent variables from the point to the entire model:

1 Switch to **Geom2** by clicking its tab.
2 Open the **Point Integration Variables** dialog box by selecting **Integration Coupling Variables>Point Variables** from the **Options** menu.
3 Enter the following names and expressions, each on a separate row in the table on the **Source** tab.

NAME	EXPRESSION
`e00`	`e0`
`e11`	`e1`
`e22`	`e2`

The variables `e00`, `e11`, and `e22` take the values of `e0`, `e1`, and `e2`, respectively, in the entire model.

SUBDOMAIN SETTINGS

Finally, enter the additional terms in the generalized plane strain problem.

1 Switch to **Geom1** by clicking its tab.
2 Select **Equation System>Subdomain Settings** from the **Physics** menu.
3 On the **Weak** tab in the **Subdomain Settings - Equation System** dialog box, add `-E/((1+nu)*(1-2*nu))*(ex_test*nu*ez+ey_test*nu*ez)-ez_test*sz` in the second row of the **weak** term for all subdomains. The two first rows correspond to the plane strain problem. You can enter this expression in any of these two fields, because the software adds them together when assembling the problem.

MESH GENERATION

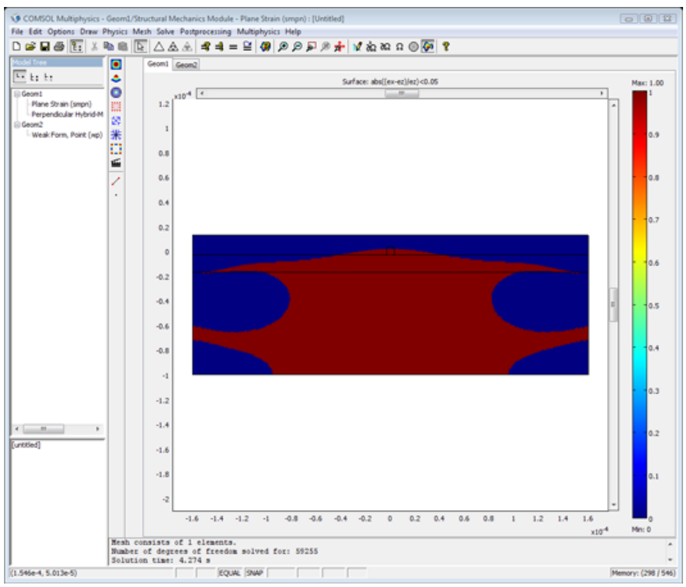
1 Initialize the mesh.
2 Refine it twice.
3 Switch to **Geom2** by clicking its tab.
4 Initialize the mesh in this geometry too.

COMPUTING THE SOLUTION

1 In the **Solver Parameters** dialog box select the **Stationary** solver. When done, click **OK**.
2 On the **Solve for** tab in the **Solver Manager** dialog box, select only the **Plane Strain** and the **Weak Form, Point** application modes. This excludes the hybrid-mode waves application mode from the equations to solve. When done, click **OK**.
3 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 Switch to **Geom1** by clicking its tab.
2 Click the **Surface** tab on the **Plot Parameters** dialog box. Change the string in the **Expression** edit field on the **Surface Data** page to `abs((ex-ez)/ez)<0.05`. Clear the **Smooth** check box. Click **OK** to visualize the area where the relative difference between the x and z strain is within 5%. The model is most accurate in the regions close to the core, far from the boundaries on the far left and right.

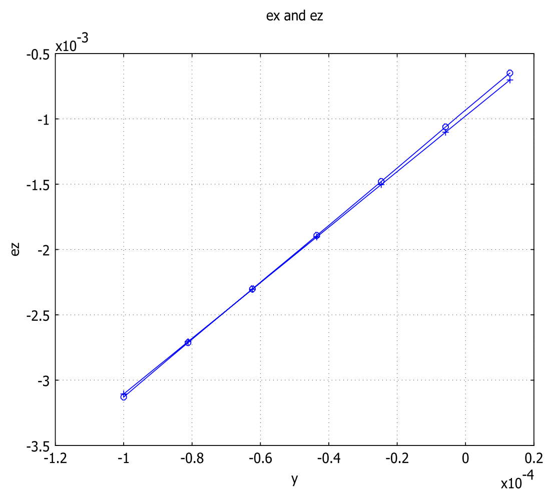


3 For symmetry reasons the strains ε_x and ε_z should be equal. To see how well this model has achieved this, make a cross-section plot of them. Open the **Cross-Section Plot Parameters** dialog box. Enter the **Title** `ex and ez` by using the **Title/Axis** dialog box on the **General** page. On the **Line/Extrusion** tab, set **x0** to `0`, **x1** to `0`, **y0** to `-1e-4`, and **y1** to `1.3e-5` to make a vertical line across the structure. Enter the **y-axis data**

expression `ex`, and set the **Line resolution** to `25`. Set the **x-axis data** to `y`. Click the **Line Settings** button and select **Circle** as **Line marker**. Click **Apply** to plot `ex`.

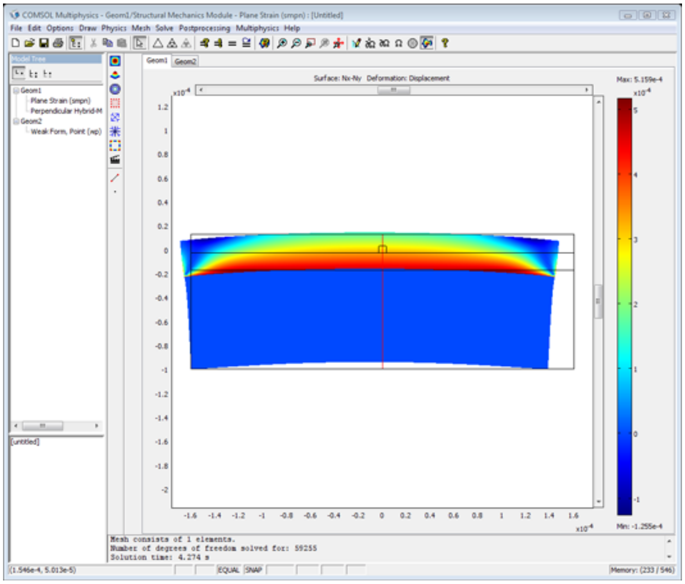
4 On the **General** tab check **Keep current plot** to be able to plot e_z in the same figure window.

5 Then change the **Expression** to `ez` in **y-axis data**. Use **Plus sign** as **Line marker** for this plot. Click **OK** to make the plot.



6 To view the stress-induced birefringence $n_x - n_y$ together with the deformation, open the **Plot Parameters** dialog box and select **Surface** and **Deformation** plot. On the **Surface** page, enter the `Nx-Ny` in the **Expression** edit field on the **Surface Data** page, and select the **Smooth** check box again. As **Deformation data** use **Displacement (smpn)**

from the **Predefined quantities** list on the **Subdomain Data** page, which is the default selected on the **Deform** tab. Click **OK**.



Note: Thid model includes an extension of the plane strain PDE system. This means that some of the postprocessing variables in the Plane Strain application mode are no longer valid and give wrong results if used. We recommend that you use only the variables entered in the **Subdomain Expressions** dialog box for postprocessing.

Optical Mode Analysis

Now continue with the mode analysis in the same way as was done in the previous model on page 608.

GEOMETRY MODELING

You can reduce the computational domain significantly for the optical mode analysis, because the energy of the fundamental modes is concentrated in the core region, and the energy density decays exponentially in the cladding and buffer regions.

Draw a rectangle with corners at (-1e-5, -1e-5) and (1e-5, 1e-5).

PHYSICS SETTINGS

Next, set up the parameters for the hybrid-mode wave problem. Select the **Perpendicular Hybrid-Mode Waves** application mode in the **Multiphysics** menu.

Scalar Variables

1. From the **Physics** menu, choose **Scalar Variables**.
2. In the **Application Scalar Variables** dialog box, set the free space wavelength to `1.55e-6`. Click **OK**.

Subdomain Settings

1. From the **Physics** menu, choose **Subdomain Settings**.
2. Deactivate the Perpendicular Hybrid-Mode Waves application mode in all subdomains except 4, 5, and 6 (clear the **Active in this domain** check box for these subdomains). This makes sure that the solution for the electromagnetic waves only includes the newly drawn rectangle.
3. For all active domains, select **n (anisotropic)** and set `Nx`, `Ny`, and `Nz`, respectively, as anisotropic refractive indices by changing the entries on the diagonal in the refractive index matrix. When done, click **OK**.

EXPRESSION VARIABLES

Occasionally the associative geometry algorithm does not update all variables and settings exactly as expected when modifying the geometry. In this case the expression for `N` might have got the wrong value in domain 6. In the **Subdomain Expressions** dialog box change the value of `N` in Subdomain 6 to `nCore`.

MESH GENERATION

1. On the **Subdomain** tab in the **Free Mesh Parameters** dialog box, set the **Maximum element size** to `2e-6` for Subdomains 4 and 5, and to `2.5e-7` for Subdomain 6.
2. Initialize the mesh.

COMPUTING THE SOLUTION

1. In the **Solver Parameters** dialog box, select the **Eigenvalue** solver.
2. Enter `1.46` in the text field **Search for effective mode indices around**. The default **Desired number of effective mode indices** is set to `6`, which is fine. These settings make the eigenmode solver search for the 6 eigenmodes with effective mode indices closest to the value `1.46`. This value is an estimate of the effective mode index for the fundamental mode. For propagating modes it must hold that

$$n_{\mathrm{eff}} < n_{\mathrm{core}} = 1.456$$

3 In the **Solver Manager** dialog box, select only the **Perpendicular Hybrid-Mode Waves** application mode on the **Solve For** page. This ensures that only this application mode is part of the eigenmode computation.

4 On the **Initial Value** page, set the initial value to **Current solution**. This takes the plane strain solution and use it to evaluate the refractive indices for the mode analysis.

5 Click the **Solve** button.

The following table contains the effective mode indices together with the effective mode indices obtained from the analysis in the previous model without the generalization of the plane strain equations.

EFFECTIVE MODE INDEX	GENERALIZED PLANE STRAIN	PLANE STRAIN	DIFFERENCE *1e-3
n_{eff1}	1.450403	1.451136	0.73
n_{eff2}	1.450115	1.450872	0.76
n_{eff3}	1.444671	1.445405	0.73
n_{eff4}	1.444398	1.445139	0.74
n_{eff5}	1.444383	1.445132	0.75
n_{eff6}	1.444109	1.444865	0.76

There is a systematic shift in the propagation constants when the strain in the z direction is taken into account.

15

Thermal-Structure Interaction

This chapter contains models that include the interaction between heat transfer and thermal expansion, often called *thermal-structure interaction*.

Thermal Stresses in a Layered Plate

Introduction

This example contains an analysis of the thermal stress in a layered plate. The plate consists of three layers: the coating, the substrate, and the carrier. The coating is deposited on the substrate at a temperature of 800 °C. At this temperature both the coating and the substrate are stress-free. The temperature of the plate is then lowered to 150 °C, which induces thermal stresses in the coating/substrate assembly. At this temperature the coating/substrate assembly is epoxied to a carrier plate so that the coating/substrate has initial stresses when it is bonded to the carrier. Finally, the temperature is lowered to 20 °C.

Model Definition

The plate is restrained from moving in the *z* direction. This makes it possible to use the 2D Plane Strain application mode, which assumes that the *z*-component of the strain is zero.

This model contains only thermal loads, and they are introduced into the constitutive equations according to the following equations:

$$\sigma = D\varepsilon_{\mathrm{el}} + \sigma_0 = D(\varepsilon - \varepsilon_{\mathrm{th}} - \varepsilon_0) + \sigma_0$$

and

$$\varepsilon_{\mathrm{th}} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix}_{\mathrm{th}} = \alpha_{\mathrm{vec}}(T - T_{\mathrm{ref}})$$

where σ is the stress vector, D is the elasticity matrix, ε_x, ε_y, ε_z, γ_{xy}, γ_{yz}, γ_{xz} are the strain components, α_{vec} is the thermal expansion coefficients, T is the actual temperature, and T_{ref} is the reference temperature. For more details concerning the governing and constitutive equations see the chapters "Thermal Strain" on page 178 and "Plane Strain" on page 112 in the *Structural Mechanics Module User's Guide*.

The geometry of the plate is shown in Figure 15-1. The top layer in the geometry is the coating, the middle layer is the substrate, and the bottom layer is the carrier.

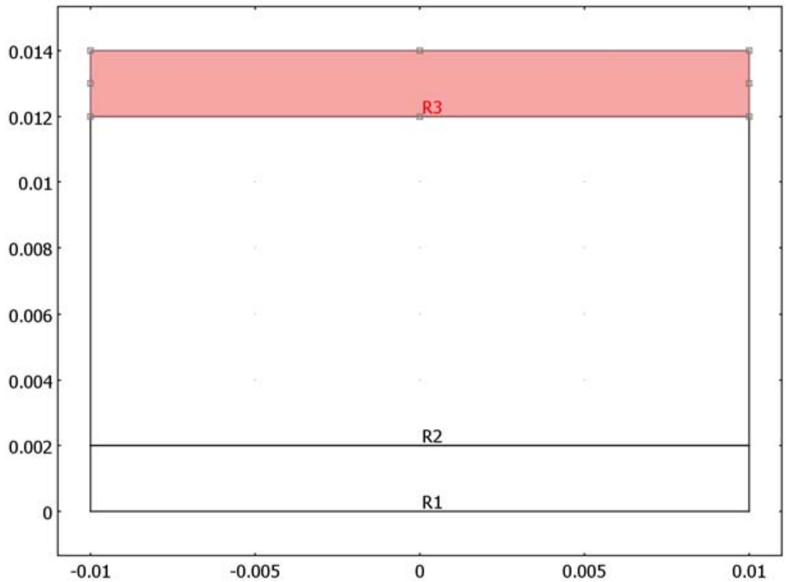


Figure 15-1: The plate geometry.

The analysis uses two steps:

STEP 1

In the first step you lower the temperature from 800 °C to 150 °C, which affects the coating layer and the substrate layer. The carrier layer is not active in this step.

The lower-left corner of the substrate is fixed, and the lower-right corner of the substrate is constrained in the *y* direction. This prevents rigid-body movements but does not affect the stress distribution.

STEP 2

In this step all three layers are active and you drop the temperature from 150 °C to room temperature, 20 °C. This step includes the initial stresses from Step 1.

In the same manner as in Step 1 the lower-left corner of the carrier is fixed, and the lower-right corner of the carrier is constrained in the *y* direction.

Results and Discussion

The normal stress in the *x* direction from the first analysis step is depicted in Figure 15-2. The substrate material has a higher thermal expansion coefficient than the coating material. This means that the substrate shrinks more than the coating, causing tensile stresses in the substrate area next to the coating and compressive stresses in the coating.

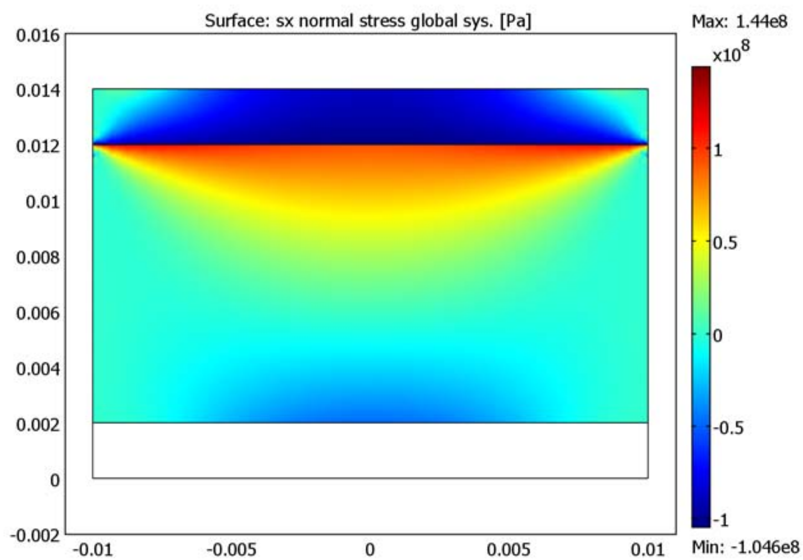


Figure 15-2: Normal stress in the x direction for the first analysis step.

Figure 15-3 shows the residual thermal *x*-stress in the final step where the temperature is lowered to 20 °C. The tensile stress levels have increased somewhat in the substrate area next to the coating, as have the compressive stress in the coating compared to the first process step. The main stress contribution is clearly the added initial stress from the first process step.

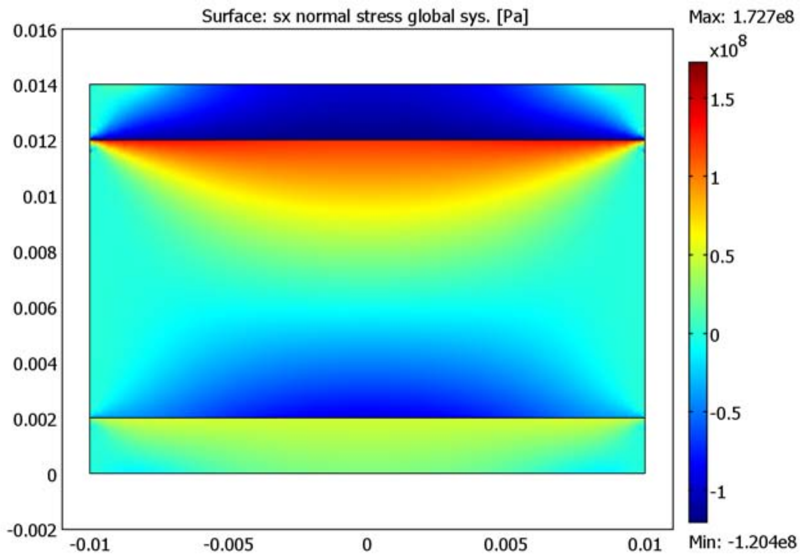


Figure 15-3: Residual thermal stress at room temperature.

Model Library path: `Structural_Mechanics_Module/Thermal-Structural_Interaction/layered_plate`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 On the **New** page select **2D** from the **Space Dimension** list and click on **Structural Mechanics Module**.

2 Select **Plane Strain>Static analysis**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Axes/Grid Settings**.

2 Specify axis and grid settings according to the following table; when done, click **OK**.

AXIS		GRID	
x min	`-1.5e-2`	x spacing	`5e-3`
x max	`1.5e-2`	Extra x	-
y min	`-2e-3`	y spacing	`2e-3`
y max	`20e-3`	Extra y	-

3 Open the **Constants** dialog box and enter the following constant names, expressions, and descriptions (optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`Ttop`	`800[degC]`	`Coating deposition temperature`
`Tbot`	`150[degC]`	`Temperature when the coating/substrate is epoxied to the carrier`
`Troom`	`20[degC]`	`Room temperature`

GEOMETRY MODELING

1 Draw the first rectangle with opposite corners at (–0.01, 0) and (0.01, 0.002).

2 Draw the second rectangle on top of the first rectangle with opposite corners at (–0.01, 0.002) and (0.01, 0.012).

3 Draw the third rectangle on top of the second rectangle with opposite corners at (–0.01, 0.012) and (0.01, 0.014).

4 See Figure 15-1 for the result.

PHYSICS SETTINGS

Point Settings

Enter point settings according to the following table.

POINT	2		6	
Type	Constraint		Constraint	
	R_x R_y	0 0	R_y	0

Subdomain Settings

The material properties are specified in the **Materials/Coefficients** dialog box.

1 Select **Materials/Coefficients Library** from the **Options** menu.

2 Click the **New** button and specify the following material parameters:

PARAMETER	EXPRESSION/VALUE	COMMENT
E	`7e10`	Young's modulus, coating
ν (nu)	`0.17`	Poisson's ratio, coating
α (alpha)	`5e-7`	Thermal expansion coefficient, coating

3 Change the **Name** to `Coating`.

4 Click **Apply**.

5 Click the **New** button specify the following second set of material parameters:

PARAMETER	EXPRESSION/VALUE	DESCRIPTION
E	`1.3e11`	Young's modulus, substrate
ν (nu)	`0.28`	Poisson's ratio, substrate
α (alpha)	`3e-6`	Thermal expansion coefficient, substrate

6 Change the **Name** to `Substrate`.

7 Click **Apply**.

8 Click the **New** button and specify the following third set of material parameters:

PARAMETER	EXPRESSION/VALUE	DESCRIPTION
E	`2.15e11`	Young's modulus, carrier
ν (nu)	`0.3`	Poisson's ratio, carrier
α (alpha)	`6e-6`	Thermal expansion coefficient, carrier

9 Change the **Name** to `Carrier`.

10 Click **OK**.

Next, specify the subdomain settings:

1 Choose **Physics>Subdomain Settings**.

2 To set the first subdomain inactive, select Subdomain 1 from **Subdomain selection** list and then clear the **Active in this subdomain** check box.

3 Enter the following subdomain settings for the remaining two subdomains:

SUBDOMAIN	2		2	
Page	Material		Load	
	Library material	Substrate	Include thermal expansion	√

SUBDOMAIN	2		2	
			Temp	`Tbot`
			Tempref	`Ttop`

SUBDOMAIN	3		3	
Page	Material		Loads	
	Library material	Coating	Include thermal expansion	√
			Temp	`Tbot`
			Tempref	`Ttop`

MESH GENERATION

1 Initialize the mesh.

2 Refine the mesh once.

MULTIPHYSICS SETTINGS

1 Open the **Model Navigator** by selecting **Model Navigator** from the **Multiphysics** menu.

2 Choose the **Structural Mechanics Module>Plane Strain>Static analysis**. Make sure that it is the Structural Mechanics Module's Plane Strain application mode.

3 Add this application mode to the model by clicking the **Add** button.

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

Check that the second Plane Strain application mode is selected in the **Multiphysics** menu.

Point Settings

Enter point constraints according to the following table.

POINT	1		5	
Type	Constraint		Constraint	
	R_x	0		
	R_y	0	R_y	0

Subdomain Settings

1 Enter subdomain settings according to the following table.

SUBDOMAIN	1		1	
Page	Material		Load	
	Library material	Carrier	Include thermal expansion	√

SUBDOMAIN	1		1	
			Temp	`Troom`
			Tempref	`Tbot`

SUBDOMAIN	2		2	
Page	Material		Load	
	Library material	Substrate	Include thermal expansion	√
			Temp	`Troom`
			Tempref	`Tbot`

SUBDOMAIN	3		3	
Page	Material		Load	
	Library material	Coating	Include thermal expansion	√
			Temp	`Troom`
			Tempref	`Tbot`

2 Select Subdomains 2 and 3 from the **Subdomain selection** list in the **Subdomain Settings** dialog box.

3 Click the **Initial Stress and Strain** tab. Add the initial stresses from the first application mode by entering `sx_smpn`, `sy_smpn`, `sz_smpn` in the **Initial normal stress** edit fields and `sxy_smpn` in the **Initial shear stress** edit field.

COMPUTING THE SOLUTION

1 Select the **Solver Manager** from the **Solve** menu.

2 Select **Plane Strain (smpn)** from the **Solve for variables** list on the **Solve For** page.

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

4 Click the **Solve** button on the Main toolbar.

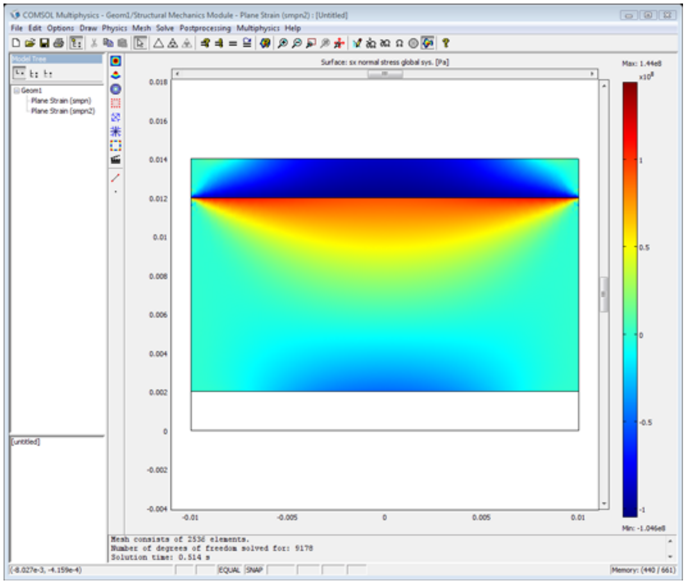
POSTPROCESSING AND VISUALIZATION

Plot the stress in *x* direction as follows:

1 Select **Plane Strain (smpn)>sx normal stress global sys.** from the list on the **Surface** page in the **Plot Parameters** dialog box.

2 Clear the **Smooth** check box.

3 Click **OK**.



COMPUTING THE SOLUTION

1 Select the **Solver Manager** from the **Solve** menu.

2 Select **Plane Strain (smpn2)** from the **Solve for variables** list on the **Solve For** page.

3 Click **OK**.

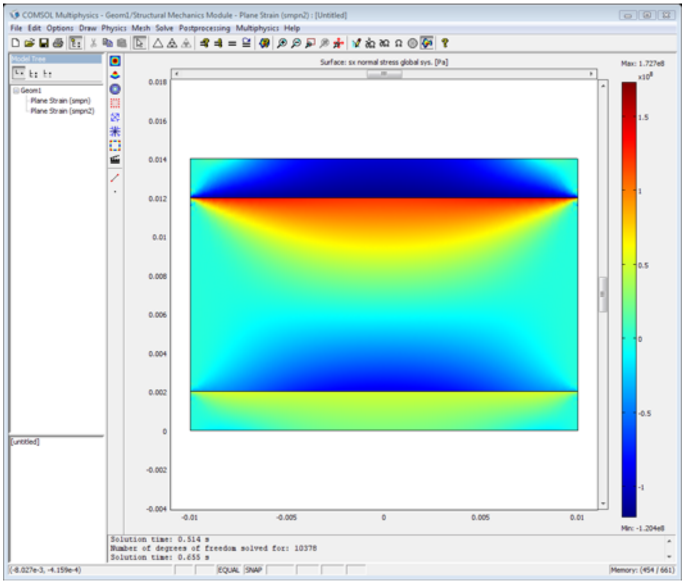
4 Click the **Restart** button.

POSTPROCESSING AND VISUALIZATION

Plot the stress in *x* direction for the second plane strain application.

1 Select **Plane Strain (smpn2)>sx normal stress global sys.** from the **Surface** page in the **Plot Parameters** dialog box and clear the **Smooth** check box.

2 Click **OK**.



COMSOL Multiphysics - Geom1/Structural Mechanics Module - Plane Strain (smpn2) : [Untitled]
File Edit Options Draw Physics Mesh Solve Postprocessing Multiphysics Help
Model Tree
Geom1
Plane Strain (smpn)
Plane Strain (smpn2)
Surface: sx normal stress global sys. [Pa]
Max: 1.727e8
Min: -1.204e8
Solution time: 0.514 s
Number of degrees of freedom solved for: 10378
Solution time: 0.655 s
EQUAL
SNAP

Surface-Mount Resistor

The drive for miniaturizing electronic devices has resulted in today's extensive use of surface-mount electronic components. An important aspect in electronics design and the choice of materials is a product's durability and lifetime. For surface-mount resistors and other components producing heat it is a well-known problem that temperature cycling can lead to cracks propagating through the solder joints, resulting in premature failure (Ref. 1). For electronics in general there is a strong interest in changing the soldering material from lead- or tin-based solder alloys to other mixtures.

The following multiphysics example models the heat transport and structural stresses and deformations resulting from the temperature distribution using the General Heat Transfer application mode and the Solid, Stress-Strain application mode.

Model Definition

Figure 15-4 shows a photograph of a surface-mount resistor together with a diagram of it on a printed circuit board (PCB).

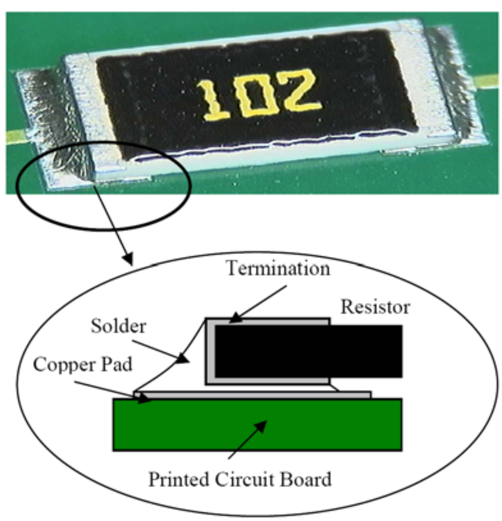


Figure 15-4: A photo and diagram of a typical surface-mounted resistor soldered to a PCB.

Table 15-1 shows the dimensions of the resistor and other key components in the model including the PCB.

TABLE 15-1: COMPONENT DIMENSIONS

COMPONENT	LENGTH	WIDTH	HEIGHT
Resistor (Alumina)	6 mm	3 mm	0.5 mm
PCB (FR4)	16 mm	8 mm	1.6 mm
Cu pad	2 mm	3 mm	35 µm
Ag termination	0.5 mm	3 mm	25 µm
Stand-off (gap to PCB)	-	-	105 µm

The simulation uses a symmetry cut along the length of the resistor so that it needs to include only half of the component (Figure 15-5).

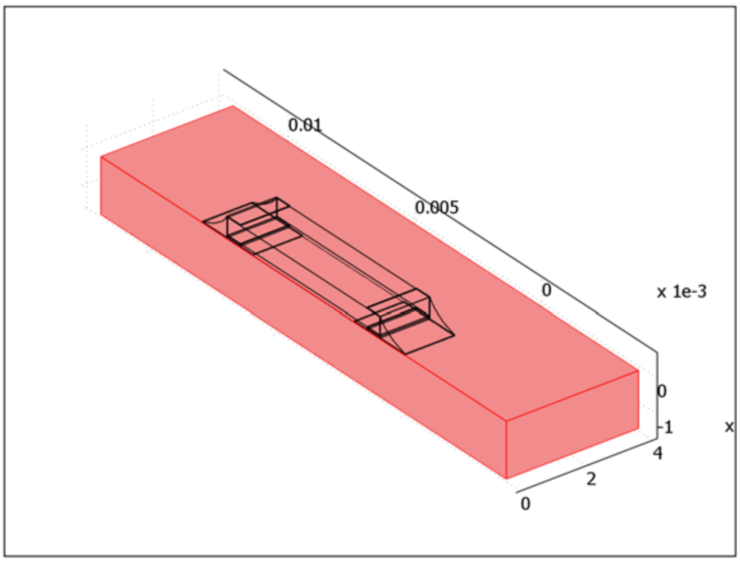


Figure 15-5: The simulation models only half the resistor.

In operation, the resistor dissipates 0.2 W of power as heat. Conduction to the PCB and convection to the surrounding air provide cooling. In this model, the heat transfer occurs through conduction in the subdomains. The model simplifies the surface cooling and describes it using a heat transfer coefficient, h, in this case set to 5 W/ $(m^2 \cdot K)$; the surrounding air temperature, T_{inf}, is at 300 K. The resulting heat-transfer equation and boundary condition (included in the model using the General Heat Transfer application mode) are

$$\nabla \cdot (-k\nabla T) = Q$$

$$-\mathbf{n} \cdot (-k\nabla T) = h(T_{\mathrm{inf}} - T)$$

where k is the thermal conductivity, and Q is the heating power per unit volume of the resistor (set to 16.7 MW/m^3 corresponding to 0.2 W in total).

The model handles thermal expansion using a static structural analysis using the Solid, Stress-Strain application mode (a description of the corresponding equations is available in the *Structural Mechanics Module User's Guide*). The thermal and mechanical material properties in this model are:

TABLE 15-2: MATERIAL PROPERTIES

MATERIAL	E (GPa)	n	α (ppm)	k (W/(m·K))	ρ (kg/m^3)	C_p (J/(kg·K))
Ag	83	0.37	18.9	420	10500	230
Alumina	300	0.222	8.0	27	3900	900
Cu	110	0.35	17	400	8700	385
Fr4	22	0.28	18	0.3	1900	1369
60Sn-40Pb	10	0.4	21	50	9000	150

The model treats properties of air as temperature dependent according to the following equations (Ref. 3):

$$\rho = (p_0 M_{\mathrm{w}})/(RT)$$

with $p_0 = 101.3$ kPa, $M_{\mathrm{w}} = 0.0288$ kg/mol, and $R = 8.314$ J/(mol·K). Further,

$$C_p = 1100 \ \ \mathrm{J/(kg{\cdot}K)}$$

$$k = 10^{-3.723 + 0.865\log(T)} \ \ \mathrm{W/(m{\cdot}K)}$$

The stresses are zero at 293 K. The boundary condition for the Solid, Stress-Strain application mode is that both ends, in the length direction of the circuit board, are fixed with respect to x, y, and z.

Note: This model requires the Heat Transfer Module and the Structural Mechanics Module.

Results and Discussion

The isosurfaces in Figure 15-6 show the temperature distribution at steady state. The highest temperature is approximately 420 K, appearing in the center of the resistor. The circuit board also heats up significantly.

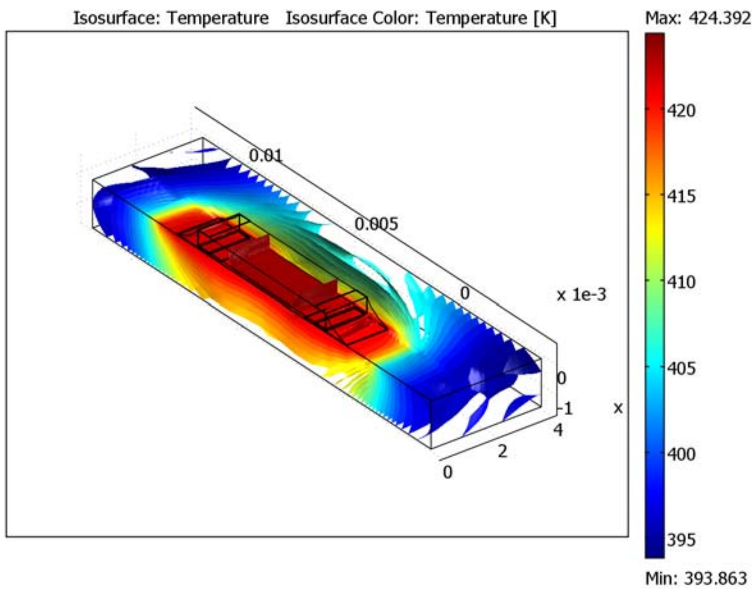


Figure 15-6: Temperature distribution in the resistor and the circuit board at steady state.

Thermal stresses appear as a result of the temperature increase; they arise from the materials' different expansion coefficients. Figure 15-7 plots the effective stress (von Mises) together with the resulting deformation of the assembly

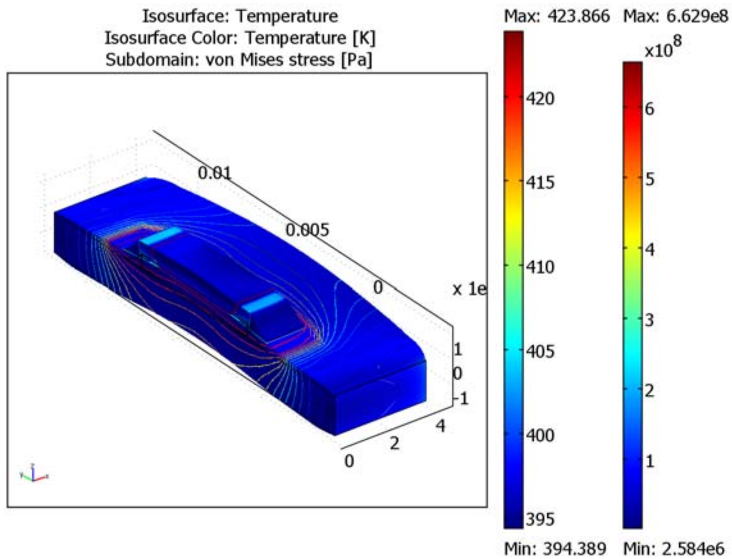


Figure 15-7: The thermally induced distribution of von Mises effective stress together with the deformation (magnified) and the isotherms.

The highest stresses seem to occur in the termination material. It is interesting to compare these effective stresses to the yield stress and thereby investigate whether or not the material is irreversibly deformed. In that case the solder is the weak point. The following graph plots the stress in the solder points alone.

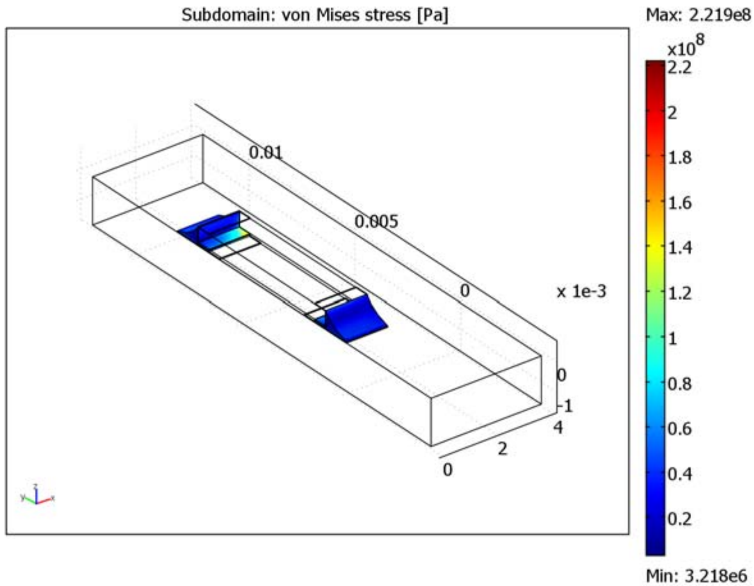


Figure 15-8: Close-up of the von Mises effective stresses in the solder joint.

The yield stress for solder is approximately 220 MPa. The highest effective stress seems to fall in the range near 220 MPa. This means that the assembly functions without failure for the tested power loads. However, if the heating power increases slightly, permanent deformation and possibly failure occur.

References

1. H.Lu, C.Bailey, M.Dusek, C.Hunt, and J.Nottay, "Modeling the Fatigue Life of Solder Joints of Surface Mount Resistors," EMAP 2000.

2. Courtesy of Dr. H. Lu, Centre for Numerical Modelling and Process Analysis, University of Greenwich, U.K.

3. J.M. Coulson and J.F. Richardson, *Chemical Engineering*, vol. 1, Pergamon Press, 1990, appendix.

Model Library path: `Structural_Mechanics_Module/Thermal-Structural_Interaction/surface_resistor`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

Open the **Model Navigator**. Click the **New** tab. In the **Space dimension** list select **2D**, then click **OK**.

OPTIONS AND SETTINGS

From the **Options** menu choose **Constants**, then define the following names and expressions. When finished, click **OK**.

NAME	EXPRESSION
T_air	293
h_air	5
q_source	0.2/(0.5e-3*3e-3*8e-3)
p	1.013e5

GEOMETRY MODELING

1 From the **Draw** menu select **Work Plane Settings**.

2 On the **Quick** page go to the **Plane** area. Click the option button for the **y-z** plane.

3 Click **OK**.

4 Create four rectangles. Shift-click the **Rectangle/Square** button on the Draw toolbar for each one and enter the data from this table:

RECTANGLE	WIDTH	HEIGHT	X	Y
R1	0.002	35e-6	0	0
R2	5.25e-4	4.5e-5	9.75e-4	3.5e-5
R3	5.25e-4	5.5e-4	9.75e-4	8e-5
R4	6e-3	5e-4	1e-3	1.05e-4

5 Click the **Zoom Extents** button on the Main menu.

6 Copy rectangle R4 by selecting it and then pressing Ctrl+C.

7 Click the **Create Composite Object** button on the Draw toolbar. (Alternatively, select **Create Composite Object** from the **Draw** menu.)

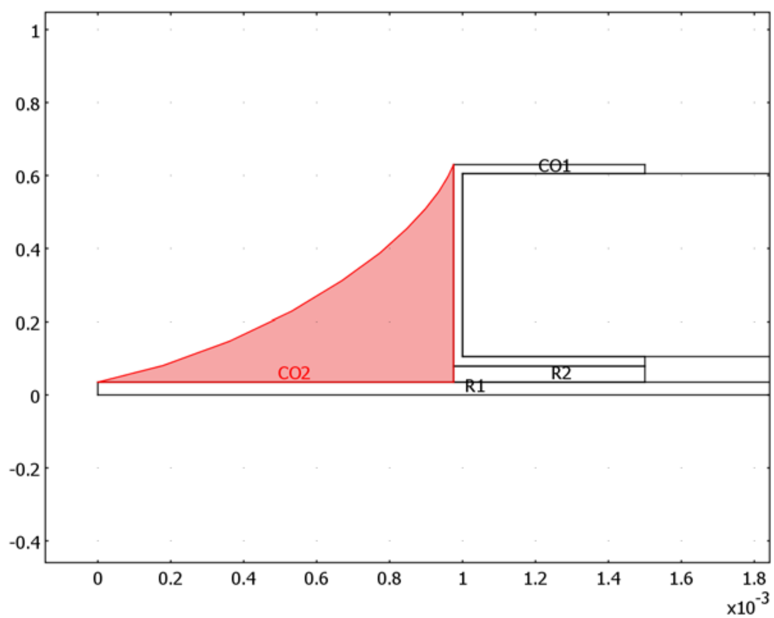
8 In the **Set formula** edit field type `R3-R4`, then click **OK**.

9 Paste a copy of R4 with zero displacement. To do so, press Ctrl+V, then click **OK** in the **Paste** dialog box.

10 Click the **2nd Degree Bezier Curve** button on the Draw toolbar.

11 Draw a curve between the upper corner of the termination and the left corner of the copper plate as in the figure below. You may want to zoom in the area before you start drawing the curve. Draw the line by clicking the coordinates $(9.75{\cdot}10^{-4}, 6.3{\cdot}10^{-4})$, $(8{\cdot}10^{-4}, 2{\cdot}10^{-4})$, and $(0, 3.5{\cdot}10^{-5})$. The coordinates that the mouse is pointing to appears in the lower left corner of the user interface.

12 After clicking the third coordinate pair click the **Line** button on the Draw toolbar. This allows you to continue the drawing with lines along the copper plate boundary and the termination boundary. Click on the coordinates $(9.75{\cdot}10^{-4}, 3.5{\cdot}10^{-5})$ and $(9.75{\cdot}10^{-4}, 6.3{\cdot}10^{-4})$. Then complete the drawing by right-clicking using the mouse. The drawing should now look like in this figure:



13 Copy the objects R1, R2, CO1, and CO2 by selecting them and pressing Ctrl+C.

14 Paste the objects by pressing Ctrl+V. Go to the **displacement** area, and in the **x** edit field type `0.006`. Click **OK**.

15 From the **Draw** menu select **Modify>Scale**. Find the **Scale factor** area, then in the **x** edit field type `-1`. Go to the **Scale base point** area and in the **x** edit field type `0.007`. Click **OK**.

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

17 Click the **Line** button on the Draw toolbar and draw a line between the coordinates (0.002, 0) and (0.006, 0).

18 To finalize the geometry select all the objects by pressing Ctrl+A and click the **Coerce to Solid** button on the Draw toolbar.

19 Shift-click the **Rectangle/Square** button. Specify settings according to the following table. When done, click **OK**.

WIDTH	HEIGHT	BASE	X	Y
16e-3	1.6e-3	Center	4e-3	-8.0e-4

20 Open the **Extrude** dialog box from the **Draw** menu.

21 From the **Objects to extrude** list select **CO5**. In the **Distance** edit field type `1.5e-3`, then click **OK**.

22 Return to the 2D geometry by clicking the **Geom1** tag.

23 Open the **Extrude** dialog box from the **Draw** menu.

24 Select **R1** from the **Objects to extrude** list and type `4e-3` in the **Distance** edit field. Click **OK**.

MESH GENERATION

1 From the **Draw** menu, select **Create Pairs**. Select both EXT1 and EXT2. Clear the **Create imprints** check box so that the meshes of the two extruded objects do not have to match at their shared boundary. This makes the mesh generation easier and reduces the number of elements.

2 Click **OK**.

3 Open the **Mapped Mesh Parameters** dialog box from the **Mesh** menu, then click the **Edge** tab.

4 Select **1** from the **Edge selection** list. Select the **Constrained edge element distribution** check box and enter `5` in the **Number of edge elements** edit field.

5 Repeat Step 4 for Edge 2, but leave the number of edge elements at its default value 10.

6 Click the **Boundary** tab and select **1** from the **Boundary selection** list.

7 Click **Mesh Selected**, then click **OK**.

8 Open the **Swept Mesh Parameters** dialog box from the **Mesh** menu.

9 Select Subdomain 1, then press Ctrl+A to select all subdomains.

10 Select the **Manual specification of element layers** check box and type 10 in the **Number of element layers** check box. Click **OK**.

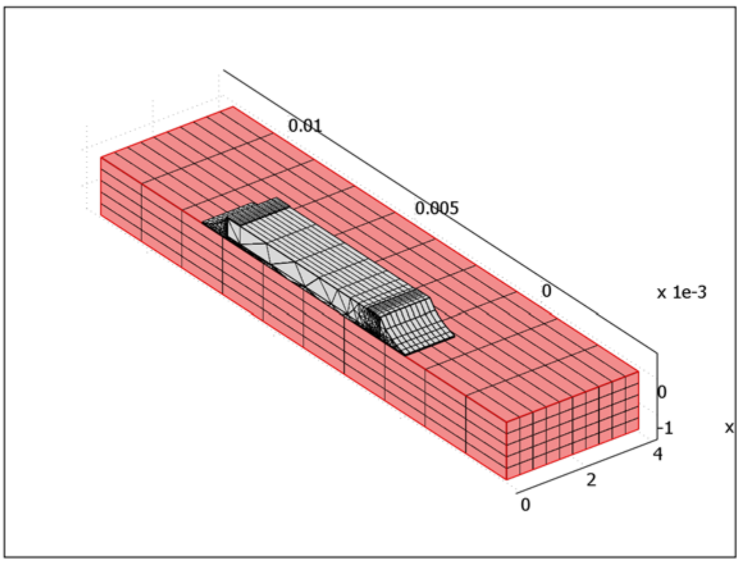
11 Click the **Subdomain mode** button on the Main toolbar.

12 Select only the PCB domain, that is, the largest subdomain.

13 Click **Mesh>Interactive Meshing>Mesh Selected (Swept)**.

14 Click **Mesh>Interactive Meshing>Mesh Remaining (Swept)**.

The meshed geometry in the drawing area should now look like that in the following figure:



PHYSICS SETTINGS

1 From the **Multiphysics** menu open the **Model Navigator**.

2 In the **Multiphysics** area on the right side of the dialog box select **Geom2**. In the list of application modes on the left select **Structural Mechanics Module>Thermal-Structural Interaction>Solid, Stress-Strain with Thermal Expansion**.

3 Click **OK**.

Subdomain Settings

1 From the **Multiphysics** menu, select **Geom2: Solid, Stress-Strain**.

2 From the **Physics** menu select **Subdomain Settings**.

3 Click the **Load** tab (not the **Load** button), then select all subdomains. In the **Tempref** edit field type `T_air`.

4 Click the **Material** tab.

5 Select Subdomain 1. Click the **Load** button to open the **Materials/Coefficients Library** dialog box. Select **Basic Material Properties>FR4 (Circuit board)**. Click **OK**.

6 Repeat the previous step for the other subdomains with materials according to the following table:

PROPERTY	SUBDOMAINS 2, 8	SUBDOMAINS 3, 4, 9, 11	SUBDOMAINS 5, 10	SUBDOMAIN 6
Material	Copper	Solder, 60Sn-40Pb	Ag	Alumina

7 Select Subdomain 7, then clear the **Active in this domain** check box.

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

9 Change the active application mode. From the **Multiphysics** menu select **Geom2: General Heat Transfer**.

10 From the **Physics** menu select **Subdomain Settings**. Go to the **Conduction** page, select Subdomain 1, and then select **FR4 (Circuit board)** from the **Library material** list.

11 Repeat for the other subdomains according to:

PROPERTY	SUBDOMAINS 2, 8	SUBDOMAINS 3, 4, 9, 11	SUBDOMAINS 5, 10	SUBDOMAIN 6	SUBDOMAIN 7
Material	Copper	Solder, 60Sn-40Pb	Ag	Alumina	Air, 1atm
Q	0	0	0	`q_source`	0

12 Go to the **Init** page. Select all subdomains and in the **Temperature** edit field, type `T_air`. Click **OK**.

Boundary Conditions

1 From the **Physics** menu open the **Boundary Settings** dialog box.

2 Select the exterior boundaries in contact with air, that is, Boundaries 3, 4, 8, 12, 19, 29, 30, 44, 46, and 52–63. In the **Boundary condition** list select **Heat flux**.

3 In the **Heat transfer coefficient** edit field type `h_air`, and in the **External temperature** edit field type `T_air`. Click **OK**.

4 In the **Multiphysics** menu change the active application mode to **Geom2: Solid, Stress-Strain**.
5 From the **Physics** menu open the **Boundary Settings** dialog box.
6 Select Boundaries 1, 7, 10, 13, 16, 20, 33, 37, 40, and 48. Select **Symmetry plane** from the **Constraint condition** list.
7 Select Boundaries 2 and 5. Select **Fixed** from the **Constraint condition** list. Click **OK**.

COMPUTING THE SOLUTION

The solution procedure runs in two steps: first solving for the temperature field, then solving for the stresses. Do so by using solver scripting to record the solver commands and then run them.

1 From the **Solve** menu open the **Solver Manager** dialog box.
2 On the **Solve For** page select **Geom2>General Heat Transfer (htgh)**. Click **Apply**.
3 In the **Script** page select the **Solve using a script** check box, then click the **Add Current Solver Settings** button.
4 On the **Solve For** page select **Geom2>Solid, Stress-Strain (smsld)**.
5 In the **Initial Value** page go to the **Values of variable not solved for and linearization point** area and select the **Current solution** button. Click **OK**.
6 From the **Solve** menu open the **Solver Parameters** dialog box.
7 In the **Linear system solver** list select **Direct (SPOOLES)** to take advantage of the symmetric system matrices. Click **OK**.
8 From the **Solve** menu select the **Solver Manager**. Click the **Script** tab, then click the **Add Current Solver Settings** button. Click **OK**.
9 Click the **Solve** button on the Main toolbar. The second part of the solution script is rather memory intensive when using a direct solver. The calculations require approximately 500 MB of free memory.

POSTPROCESSING AND VISUALIZATION

To reproduce the temperature plot in Figure 15-6:

1 From the **Postprocessing** menu open the **Plot Parameters** dialog box.
2 Click the **General** tab.
3 In the **Plot type** area clear the **Slice** check box and select the **Isosurface** check box.
4 Click the **Isosurface** tab.
5 From the **Predefined quantities** list select **General Heat Transfer (htgh)>Temperature**.

6 In the **Isosurface levels** area click the **Levels** button, then type 30 in the **Number of levels** edit field. Click the **Color Data** tab and select the **Color data** check box. From the **Predefined quantities** list select **General Heat Transfer (htgh)>Temperature**.

7 Click **Apply**.

8 Click the **Scenelight** button on the **Camera** toolbar to finish off the plot.

To reproduce Figure 15-7:

1 While still on the **Isosurface** page in the **Plot Parameters** dialog box, change the **Number of levels** to 15.

2 In the **Fill style** list select **Wireframe**.

3 Click the **Subdomain** tab and enable this plot type by selecting the **Subdomain plot** check box at the top of the dialog box. From the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>von Mises stress**.

4 Click the **Deform** tab and select the **Deformed shape plot** check box.

5 In **Domain types to deform** area clear the **Boundary** and **Edge** check boxes.

6 In the **Deformation data** area click the **Subdomain Data** tab, then select **Solid, Stress-Strain (smsld)>Displacement** from the **Predefined quantities** list.

7 Click **OK**.

To reproduce Figure 15-8:

1 From the **Options** menu select **Suppress>Suppress Subdomains**.

2 Select Subdomains 1, 2, 5, 6, 7, 8, and 10, then click **OK**.

3 From the **Postprocessing** menu open the **Plot Parameters** dialog box.

4 In the **General** page, go to the **Plot type** area, clear the **Isosurface** and **Deform** check boxes. Click **OK** to generate the plot in Figure 15-8.

Heating Circuit

Introduction

Small heating circuits find use in many applications. For example, in manufacturing processes they heat up reactive fluids. Figure 15-9 illustrates a typical heating device for this application. The device consists of an electrically resistive layer deposited on a glass plate. The layer causes Joule heating when a voltage is applied to the circuit. The layer's properties determine the amount of heat produced.

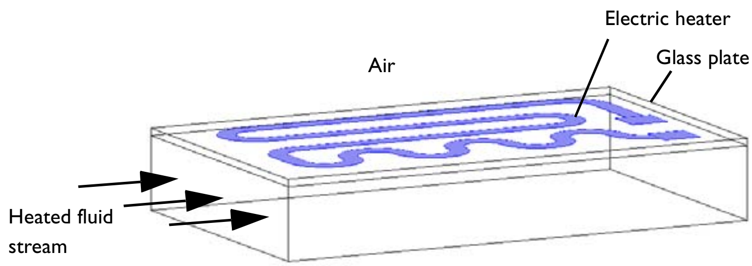


Figure 15-9: Geometry of a heating device.

In this particular application, you must observe three important design considerations:

- Non-invasive heating
- Minimal deflection of the heating device
- Avoidance of overheating the process fluid

The heater must also work without failure. You achieve the first and second requirements by inserting a glass plate between the heating circuit and the fluid; it acts as a conducting separator. Glass is an ideal material for both these purposes because it is nonreactive and has a low thermal-expansion coefficient.

You must also avoid overheating due to the risk of self-ignition of the reactive fluid stream. Ignition is also the main reason for separating the electrical circuit from direct contact with the fluid. The heating device is tailored for each application, making virtual prototyping very important for manufacturers.

For heating circuits in general, detachment of the resistive layer often determines the failure rate. This is caused by excessive thermally induced interfacial stresses. Once the layer has detached, it gets locally overheated, which accelerates the detachment. Finally, in the worst case, the circuit might overheat and burn. From this perspective, it is also important to study the interfacial tension due to the different thermal-expansion coefficients of the resistive layer and the substrate as well as the differences in temperature. The geometric shape of the layer is a key parameter to design circuits that function properly. You can investigate all of the above-mentioned aspects by modeling the circuit.

This multiphysics example simulates the electrical heat generation, the heat transfer, and the mechanical stresses and deformations of a heating circuit device. The model uses the General Heat Transfer application mode of the Heat Transfer module in combination with the Shell, Conductive Media DC application mode from the AC/DC Module and the Solid, Stress-Strain and Shell application modes from the Structural Mechanics Module.

Note: This model requires the AC/DC Module, the Heat Transfer Module, and the Structural Mechanics Module.

Model Definition

Figure 15-10 shows a drawing of the modeled heating circuit.

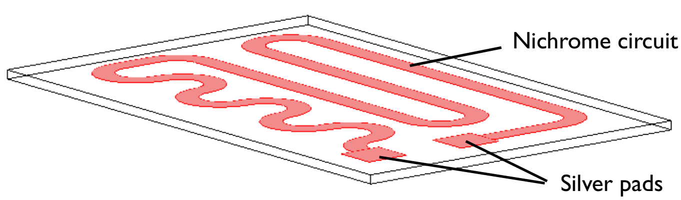


Figure 15-10: Drawing of the heating circuit deposited on a glass plate.

The device consists of a serpentine-shaped Nichrome resistive layer, 10 µm thick and 5 mm wide, deposited on a glass plate. At each end, it has a silver contact pad measuring 10 mm × 10 mm × 10 µm. When the circuit is in use, the deposited side of the glass plate is in contact with surrounding air, and the back side is in contact with the heated fluid. Assume that the edges of the glass plate are thermally insulated.

Table 15-3 gives the resistor's dimensions.

TABLE 15-3: DIMENSIONS

OBJECT	DIMENSION	SIZE
glass plate	length	130 mm
	width	80 mm
	thickness	2 mm
pads and circuit	thickness	10 µm

During operation the resistive layer produces heat. Model the electrically generated heat using the Shell, Conductive Media DC application mode from the AC/DC Module. The governing equation is

$$\nabla_t \cdot (-d\sigma\nabla_t V) = 0$$

where d is the thin layer's thickness (m), σ is the electric conductivity (S/m), V is the electric potential (V), and ∇_t denotes the gradient operator in the tangential directions. An actual applies 12 V to the pads. In the model you achieve this effect by setting the potential at one edge of the first pad to 12 V and that of one edge of the other pad to 0 V.

To model the heat transfer in the thin conducting layer, use the Highly Conductive Layer feature of the General Heat Transfer application mode. It is then not necessary to add a separate application mode for it.

The heat power per unit area (measured in W/m^2) produced inside the thin layer is given by

$$q_{\text{prod}} = dQ_{\text{DC}} \tag{15-1}$$

where $Q_{\text{DC}} = \mathbf{J}\cdot\mathbf{E} = \sigma|\nabla_t V|^2$ (W/m^3) is the power density. The generated heat appears as an inward heat flux at the surface of the glass plate.

At steady state, the resistive layer dissipates the heat it generates in two ways: on its up side to the surrounding air (at 293 K), and on its down side to the glass plate. The glass plate is similarly cooled in two ways: on its circuit side by air, and on its back side by a process fluid (353 K). You model the heat fluxes to the surroundings using heat transfer film coefficients, h. For the heat transfer to air, $h = 5$ $W/(m^2{\cdot}K)$, representing natural convection. On the glass plate's back side, $h = 20$ $W/(m^2{\cdot}K)$, representing convective heat transfer to the fluid. The sides of the glass plate are insulated.

The resulting heat transfer equation for the device, together with the boundary condition used to describe the heat fluxes at the front and back sides, is

$$\nabla \cdot (-k\nabla T) = 0$$

$$-\mathbf{n} \cdot (-k\nabla T) = q_0 + h(T_{\text{inf}} - T) - \nabla_{\text{t}} \cdot (-d_s k_s \nabla_{\text{t}} T)$$

where $\mathbf{n}$ is the normal vector of the boundary, k is the thermal conductivity (W/(m·K)), h is the heat transfer film coefficient (W/(m^2·K)), and T_{inf} is the temperature (K) of the surrounding medium. The last term on the right-hand side represents the additional flux given by the thin conducting layer, and the constant k_s is the thermal conductivity in the layer (W/(m·K)). This term is only present on the boundaries where the layer is present. Similarly, the inward heat flux, q_0, is equal to q_{prod} (see Equation 15-1) at the layer but vanishes elsewhere.

The model simulates thermal expansion using static structural-mechanics analyses. It uses the Solid, Stress-Strain application mode for the glass plate, and the Shell application mode for the circuit layer. The equations of these two application modes are described in the *Structural Mechanics Module User's Guide*. The stresses are set to zero at 293 K. You determine the boundary conditions for the Solid, Stress-Strain application mode by fixing one corner with respect to x-, y-, and z-displacements and rotation.

Table 15-4 summarizes the material properties used in the model.

TABLE 15-4: MATERIAL PROPERTIES

MATERIAL	E [GPa]	ν	α [ppm]	k [W/(m·K)]	ρ [kg/m^3]	C_p [J/(kg·K)]
Silver	83	0.37	18.9	420	10500	230
Nichrome	213	0.33	10.0	15	9000	20
Glass	73.1	0.17	55	1.38	2203	703

Results and Discussion

Figure 15-11 shows the heat that the resistive layer generates.

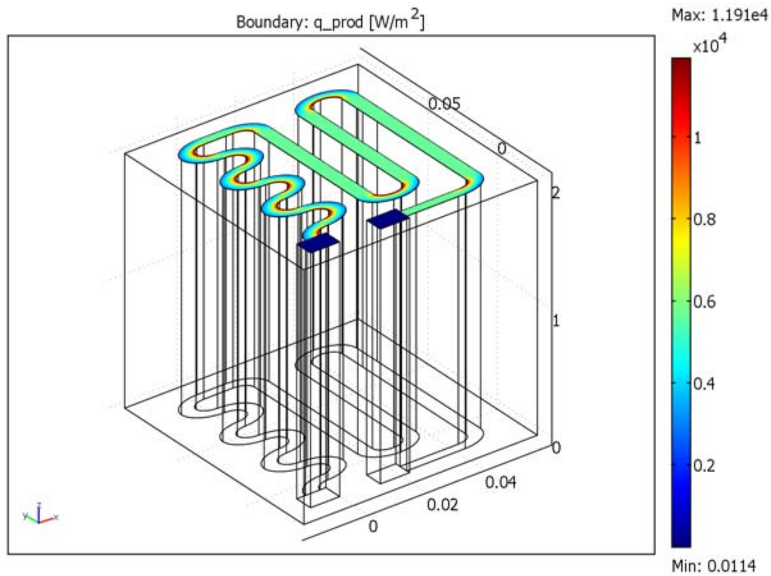


Figure 15-11: Stationary heat generation in the resistive layer when 12 V is applied.

The highest heating power arises in the inner corners of the curves due to the higher current density at these spots. The total generated heat, as calculated by integration, is approximately 13.8 W.

Figure 15-12 shows the temperature of the resistive layer and the glass plate at steady state.

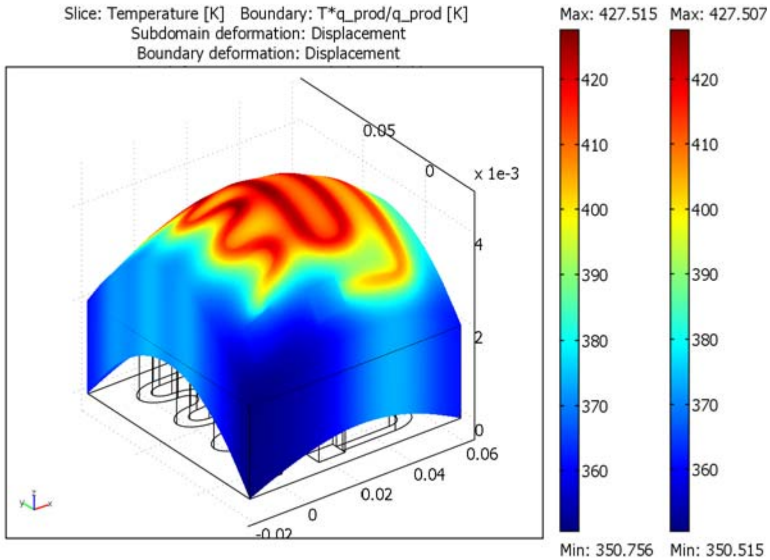


Figure 15-12: Temperature distribution in the heating device at steady state.

The highest temperature is approximately 430 K, and it appears in the central section of the circuit layer. It is interesting to see that the differences in temperature between the fluid side and the circuit side of the glass plate are quite small because the plate is very thin. Using boundary integration, the integral heat flux on the fluid side evaluates to approximately 8.5 W. This means that the device transfers the majority of the heat it generates—8.5 W out of 13.8 W—to the fluid, which is good from a design perspective, although the thermal resistance of the glass plate results in some losses.

The temperature rise also induces thermal stresses due the materials' different coefficients of thermal expansion. As a result, mechanical stresses and deformations arise in the layer and in the glass plate. Figure 15-13 shows the effective stress distribution in the device and the resulting deformations. During operation, the glass plate bends towards the air side.

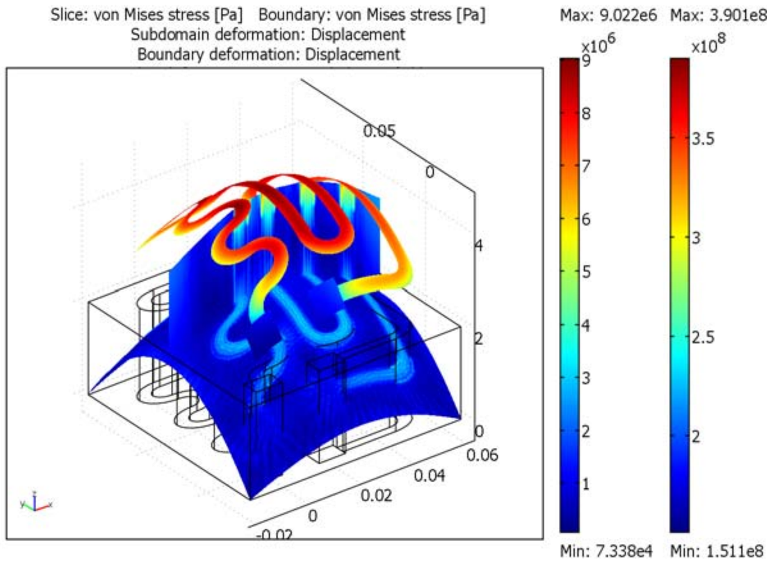


Figure 15-13: The thermally induced von Mises effective stress plotted with the deformation.

The highest effective stress, approximately 7 MPa, appears in the corners of the silver pads. The yield stress for high quality glass is roughly 250 MPa, and for Nichrome it is 360 MPa. This means that the individual objects remain structurally intact for the simulated heating power loads.

You must also consider stresses in the interface between the resistive layer and the glass plate. Assume that the yield stress of the surface adhesion in the interface is in the region of 50 MPa—a value significantly lower than the yield stresses of the other materials in the device. If the effective stress increases above this value, the resistive layer will locally detach from the glass. Once it has detached, heat transfer is locally impeded, which can lead to overheating of the resistive layer and eventually cause the device to fail.

Figure 15-14 displays the effective forces acting on the adhesive layer during heater operation. As the figure shows, the device experiences a maximum interfacial stress that is an order of magnitude smaller than the yield stress. This means that the device will be OK in terms of adhesive stress.

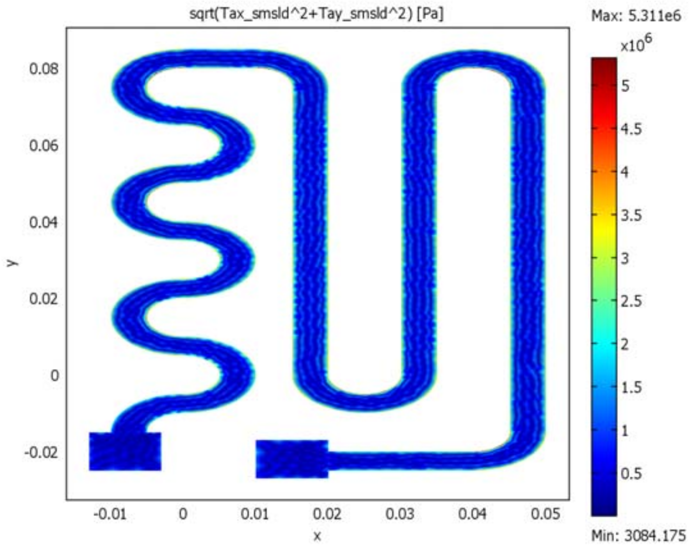


Figure 15-14: The effective forces in the interface between the resistive layer and the glass plate.

Finally study the device's deflections, depicted in Figure 15-15.

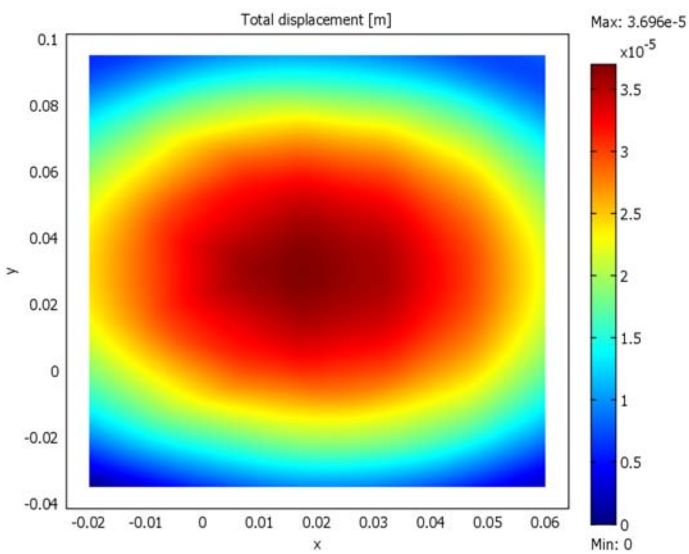


Figure 15-15: Total displacement on the fluid side of the glass plate.

The maximum displacement, located at the center of the plate, is approximately 30 µm. For high-precision applications, such as semiconductor processing, this might be a significant value that limits the device's operating temperature.

Model Library path: `Structural_Mechanics_Module/Thermal-Structure_Interaction/heating_circuit`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Open the **Model Navigator**.
2 In the **Space dimension** list select **3D**. Click the **Multiphysics** button.
3 From the list of application modes select **AC/DC Module>Statics>Shell, Conductive Media DC**. In the **Application mode name** edit field type `DC`, then click **Add**.
4 From the list of application modes select **Heat Transfer Module>General Heat Transfer**, then click **Add**.
5 Similarly add two more application modes: **Structural Mechanics Module>Solid, Stress-Strain** and **Structural Mechanics Module>Shell**. When done, click **OK**.

OPTIONS AND SETTINGS

From the **Options** menu select **Constants**. Define the following names, expressions, and descriptions (the descriptions are optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
V_in	12[V]	Input voltage
d_layer	10[um]	Layer thickness
sigma_Silver	6.3e7[S/m]	Electric conductivity of silver
sigma_Nichrome	9.3e5[S/m]	Electric conductivity of Nichrome
T_ref	293[K]	Reference temperature
T_air	T_ref	Air temperature
h_air	5[W/(m^2*K)]	Heat transfer film coefficient, air
T_fluid	353[K]	Fluid temperature
h_fluid	20[W/(m^2*K)]	Heat transfer film coefficient, fluid

GEOMETRY MODELING

1 Create a 2D work plane at $z = 0$ by first choosing **Draw>Work-Plane Settings** and then clicking **OK** in the dialog box that appears to accept the default settings.

2 Create nine rectangles. Open the **Draw>Specify Objects>Rectangle** dialog box, and for each one enter the appropriate data from this table:

RECTANGLE	WIDTH	HEIGHT	X-BASE	Y-BASE
R1	0.08	0.13	-0.02	-0.035
R2	0.01	0.02	-0.01	-0.01
R3	0.005	0.075	0.015	0
R4	0.01	0.005	0	0.08
R5	0.01	0.01	0	0.075
R6	0.005	0.015	0.045	0.06
R7	0.02	0.005	0.02	-0.025
R8	0.01	0.01	0.01	-0.027
R9	0.01	0.01	-0.013	-0.025

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

4 Create a circle with the menu item **Draw>Specify Objects>Circle**. In the **Radius** edit field type `0.01`, then click **OK**.

5 Create another circle the same way bit with a **Radius** of `0.005`.

6 Click the **Create Composite Object** button on the Draw toolbar. In the **Set formula** edit field type `C1-C2-R2`, then click **OK**. This step generates the composite object CO1.

7 Select CO1 by clicking on it, click the **Mirror** button on the Draw toolbar, then click **OK**. This step creates a mirror copy of CO1 called CO2.

8 Select CO2 and then click the **Move** button on the Draw toolbar. Go to the **Displacement** area, and in the **y** edit field type `0.015`. Click **OK**.

9 Select both CO1 and CO2 by pressing Ctrl while clicking on the objects. Copy both by pressing Ctrl+C.

10 Paste twice by pressing Ctrl+V, specifying the displacement, and clicking **OK**. For the first copy specify the **y-displacement** as `0.030`, and for the second specify `0.060`.

11 Select CO1, copy it, and paste it with zero displacement.

12 Click the **Rotate** button on the Draw toolbar. In the **Rotation angle** edit field type `-90`, then click **OK**.

13 Click the **Move** button on the Draw toolbar. In the **x** edit field type `0.025`, then click **OK**.

14 Click the **Mirror** button on the Draw toolbar. In the **Normal vector** edit field for **x** type `0`, and in the **y** edit field type `1`. Click **OK**.

15 Click the **Move** button on the Draw toolbar. In the **x** edit field type `-0.015`, and in the **y** edit field type `0.075`. Click **OK**.

16 Click the **Create Composite Object** button on the Draw toolbar. In the **Set formula** edit field type `CO8-R5`, then click **OK**. This step generates composite object CO9.

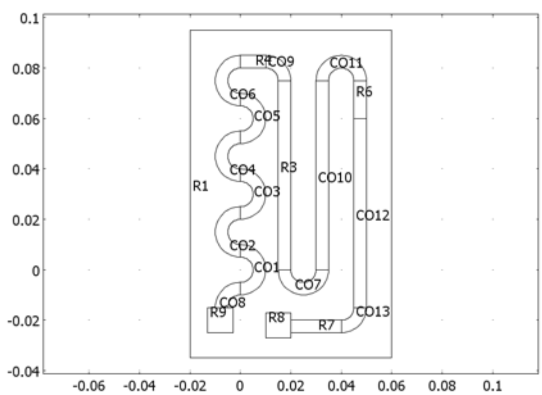
17 Copy and paste CO9 with **x-** and **y-displacements of** `-0.02` and `-0.08`, respectively.

18 Click the **Rotate** button on the Draw toolbar. In the α edit field in the **Rotation angle** area type `90`, and in the **y** edit field in the **Center point** area type `-0.005`. Click **OK**.

19 Select objects CO7 and R3. Click the **Mirror** button on the Draw toolbar. In the **Normal vector** edit field for **x** type `0`, and in the **y** edit field type `1`. Click **OK**.

20 Click the **Move** button on the Draw toolbar. Specify the displacement by typing `0.015` in the **x** edit field and `0.075` in the **y** edit field, then click **OK**. These steps generate composite objects CO10 and CO11.

21 Select objects CO9 and R3. Using the **Move** dialog box, repeat the procedure in the previous two steps with values for the **x-** and **y-displacements** of `0.03` and `0.06`. The geometry should now look like that in the following figure.



22 Select all objects except the glass plate (R1) and the silver tabs (R8 and R9). Click the **Create Composite Object** button on the Draw toolbar. Clear the **Keep interior boundaries** check box, then click **OK**. This step generates composite object CO14.

23 Select CO14, R8, and R9, then click the **Coerce to Solid** button on the Draw toolbar.

MESH GENERATION

1 From the **Mesh** menu open the **Free Mesh Parameters** dialog box.

2 On the **Global** page go to the **Predefined mesh sizes** list and select **Coarse**.

3 On the **Subdomain** page select Subdomain 3, then in the **Maximum element size** edit field type `2e-3`.

4 Click the **Remesh** button, then click **OK**.

5 From the **Mesh** menu open the **Extrude Mesh** dialog box. On the **Geometry** page find the **Distance** edit field and type `2e-3`. From the **Extrude to geometry** list select **Geom1**.

6 Click the **Mesh** tab. In the **Number of element layers** edit field type `2`, then click **OK**.

7 Double-click the **EQUAL** button on the status bar at the bottom of the user interface, then click the **Zoom Extents** button on the Main toolbar to expand the geometry's z-axis.

PHYSICS SETTINGS

1 From the **Options** menu open the **Materials/Coefficients Library** dialog box.

2 Set up the materials silver and NiChrome. To do so, click **New**, then enter the settings from the following table in the corresponding edit fields. When done, click **OK**.

NAME	C	E	alpha	k	nu	rho
`Silver`	`230`	`83e9`	`18.9e-6`	`420`	`0.37`	`10500`
`Nichrome`	`230`	`213e9`	`10e-6`	`15`	`0.33`	`9000`

3 Choose **Options>Expressions>Scalar Expressions**. In the **Name** edit field type `q_prod`, and in the **Expression** edit field type `d_layer*Q_DC`. Enter `Heat power per unit area inside thin layer` in the **Description** edit field (optional). Click **OK**.

Boundary Settings—Shell, Conductive Media DC (DC)

1 From the **Multiphysics** menu select **Shell, Conductive Media DC (DC)**.

2 From the **Physics** menu select **Boundary Settings**.

3 Select all the boundaries, then clear the **Active in this domain** check box.

4 Select Boundary 14, then click the **Active in this domain** check box. In the **Electric conductivity** edit field for **σ (isotropic)** type `sigma_Nichrome`, and in the **Layer thickness** edit field type `d_layer`.

5 Repeat the previous step for Boundaries 9 and 47 but in the **Electric conductivity** edit field type `sigma_Silver`. Click **OK**.

6 From the **Physics** menu select **Edge Settings**.

7 Select Edge 13. In the **Boundary condition** list select **Electric potential**, then in the **Electric potential** edit field type `V_in`.

8 Select Edge 109. In the **Boundary condition** list select **Ground**, then click **OK**.

Subdomain Settings—General Heat Transfer

1 From the **Multiphysics** menu select **General Heat Transfer (htgh)**.

2 From the **Physics** menu open the **Subdomain Settings** dialog box, then select all the subdomains.

3 Go to the **Conduction** page. Click the **Load** button. From the **Materials** list select **Library1>Silica Glass**, then click **OK**.

4 Go to the **Init** page, and in the **$T(t_0)$** edit field type `T_ref`. Click **OK**.

Boundary Conditions—General Heat Transfer

1 From the **Physics** menu open the **Boundary Settings** dialog box. Select Boundaries 9, 14, and 47.

2 Click the **Highly Conductive Layer** tab. Select the **Enable heat transfer in highly conductive layer** check box, then in the **d_s** edit field type `d_layer`.

3 Select Boundary 14. In the **Library material** list select **Nichrome**.

4 Similarly, for Boundaries 9 and 47 select **Silver**.

5 Click the **Boundary Condition** tab. Select Boundaries 9, 14, and 47. In the **Boundary condition** list select **Heat flux**. In the **q_0** edit field type `q_prod`, in the **h** edit field type `h_air`, and in the **T_{inf}** edit field type `T_air`.

6 Repeat the settings in the previous step for Boundary 4 but without specifying **q_0**.

7 Select Boundaries 3, 8, 13, and 46. In the **Boundary condition** list select **Heat flux**. In the **h** edit field type `h_fluid`, and in the **T_{inf}** edit field type `T_fluid`. Click **OK**.

Subdomain Settings—Solid, Stress-Strain

1 From the **Multiphysics** menu select **Solid, Stress-Strain (smsld)**.

2 From the **Physics** menu open the **Subdomain Settings** dialog box, then select all the subdomains.

3 Go to the **Material** page. In **Library material** list select **Silica Glass**.

4 Click the **Load** tab. Select the **Include thermal expansion** check box. In the **Temp** edit field type `T` and in the **Tempref** edit field type `T_ref`.

5 Go to the **Element** page. In the **Predefined elements** list select **Lagrange - Linear**. Click **OK**.

Point Settings—Solid, Stress-Strain

1 From the **Physics** menu open the **Point Settings** dialog box.
2 Select Point 1. Select the check boxes next to $\mathbf{R_x}$, $\mathbf{R_y}$, and $\mathbf{R_z}$.
3 Select Point 3, then select the $\mathbf{R_z}$ check box.
4 Select Point 125, then select the $\mathbf{R_y}$ and $\mathbf{R_z}$ check boxes. Click **OK**.

Boundary Settings—Shell

1 From the **Multiphysics** menu select **Shell (smsh)**.
2 From the **Physics** menu open the **Boundary Settings** dialog box.
3 Select all the boundaries, then clear the **Active in this domain** check box.
4 Select Boundaries 9, 14, and 47. Select the **Active in this domain** check box.
5 In the **thickness** edit field type `d_layer`.
6 Go to the **Load** page. Select the **Include thermal expansion** check box. In the **Temp** edit field type `T` and in the **Tempref** edit field type `T_ref`.
7 Click the **Material** tab. Select Boundaries 9 and 47. In the **Library material** list select **Silver**.
8 Similarly, for Boundary 14 select **Nichrome**. Click **OK**.

COMPUTING THE SOLUTION

This model is best solved using a script. Follow these steps to create the script and solve the model.

1 From the **Solve** menu open the **Solver Manager**.
2 On the **Solve For** page select **Geom1 (3D)>Shell, Conductive Media DC (DC)** and **Geom1 (3D)>General Heat Transfer (htgh)**, then click **Apply**.
3 Go to the **Script** page. Select the **Solve using a script** check box. Then click the **Add Current Solver Settings** button to generate the first half of the script.
4 From the **Solve** menu, choose **Solver Parameters**.
5 In the **Linear system solver** list select **Direct (SPOOLES)** to use this solver's ability to utilize the symmetric system matrices. Click **OK**.
6 Return to the **Solver Manager**. Go to the **Initial Value** page, then to the **Initial value** area, and click the **Current solution** option button.
7 Go to the **Solve For** page. Select **Geom1>Solid, Stress-Strain (smsld)** and **Geom1>Shell (smsh)**. Click **Apply**.

8 On the **Script** page click the **Add Current Solver Settings** button to generate the second half of the script. Click **OK** to close the **Solver Manager**.

9 Finally, click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

Generate Figure 15-11 as follows:

1 From the **Postprocessing** menu open the **Plot Parameters** dialog box.

2 On the **General** page clear the **Slice** check box, then select the **Boundary** check box.

3 Go to the **Boundary** page. In the **Expression** edit field type `q_prod`. Click **OK**.

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

To calculate the total heat generated in the circuit, follow these steps:

1 From the **Postprocessing** menu open the **Boundary Integration** dialog box.

2 Select Boundaries 9, 14, and 47. In the **Expression** edit field type `q_prod`. Click **OK**.

The calculated value, roughly 13.8 W, appears in the message log at the bottom of the graphical user interface.

Generate Figure 15-12 by executing these instructions:

1 From the **Postprocessing** menu open the **Plot Parameters** dialog box. On the **General** page go to the **Plot type** area and select the **Slice**, **Boundary**, and **Deformed shape** check boxes.

2 Go to the **Slice** page, and in the **Predefined quantities** list select **Temperature (htgh)**. In the **Slice positioning** area find the **x levels** edit field and type `0`, and in the **y levels** edit field type `1`.

3 Click the option button for **Vector with coordinates** associated with **z levels**, then in the corresponding edit field type `0`.

4 On the **Boundary** page find the **Expression** edit field and type `T*q_prod/q_prod`.

The use of `q_prod/q_prod` makes the expression for the temperature valid on the resistive layer boundary only, which is the desired effect.

5 On the **Deform** page, go to the **Deformation data** area and click the **Subdomain Data** tab. In the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>Displacement**.

6 While still in the **Deformation data** area, click the **Boundary Data** tab. In the **Predefined quantities** list select **Shell (smsh)>Displacement**. Click **Apply**.

Calculate the total heat flux to the fluid in the following way:

1 From the **Postprocessing** menu open the **Boundary Integration** dialog box.

2 Select Boundaries 3, 8, 13, and 46. In the **Expression** edit field type `h_fluid*(T-T_fluid)`, then click **OK**.

A value for the total heat flux of approximately 8.47 W appears in the message log.

To generate Figure 15-13 follow these steps:

1 While still in the **Plot Parameters** dialog box, go to the **Slice** page. In the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>von Mises stress**.

2 Click the **Boundary** tab. In the **Predefined quantities** list select **Shell (smsh)>von Mises stress**. Click **Apply**.

Figure 15-14 is obtained by executing the following instructions:

1 From the **Postprocessing** menu select **Domain Plot Parameters**.

2 On the **Surface** page select Boundaries 9, 14, and 47. In the **Expression** edit field type `sqrt(Tax_smsld^2+Tay_smsld^2)`. Click **Apply**.

This gives a plot of the norm of the surface traction vector (N/m^2) in the surface plane,

$$\begin{bmatrix} \mathrm{Ta}_x \\ \mathrm{Ta}_y \end{bmatrix} = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \end{bmatrix} \begin{bmatrix} 0 \\ 0 \\ 1 \end{bmatrix} = \begin{bmatrix} \tau_{xz} \\ \tau_{yz} \end{bmatrix}$$

Finally, to obtain Figure 15-15, proceed as follows:

1 Still on the **Surface** page of the **Domain Plot Parameters** dialog box, select Boundaries 3, 8, 13, and 46.

2 In the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>Total displacement**, then click **OK**.

Simulation of a Microrobot Leg

Introduction

This example describes the modeling of one of the legs of a silicon microrobot. Figure 15-16 below shows the microrobot leg.

The microrobot uses a technique based on polyimide V-groove joints to get each of the legs to move. The polyimide has a relatively high coefficient of thermal expansion α, which causes the leg to bend slightly when the polyimide is heated. Putting several V-grooves on each leg provides sufficient deflection (see Figure 15-16).

See Ref. 1 for a more detailed description of the microrobot.

Note: This model requires the Heat Transfer Module and the Structural Mechanics Module.

Model Definition

The model is a transient heat transfer analysis combined with a quasi-static thermal deformation analysis.

The materials used in the microrobot leg are: Si, SiN, Al, P, SiO, and pSi. They are assembled as shown below:

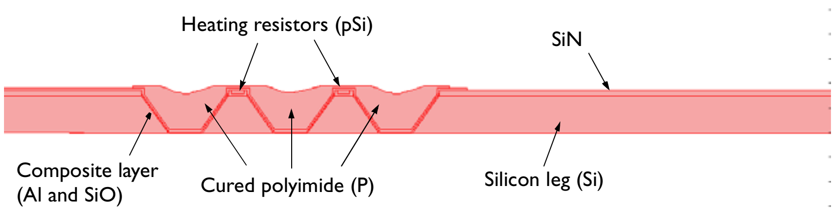


Figure 15-16: Cross-section cut of the microrobot leg.

The geometry of the model comes from an initial 2D geometry using mesh extrusion.

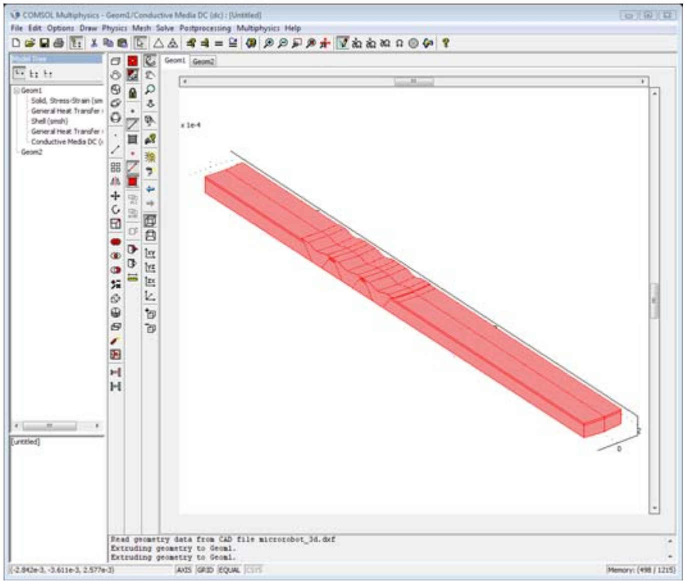


Figure 15-17: 3D extruded mesh of the microbot leg.

The heat is generated by resistive heating. An electric potential of 30 mV is applied in each of two heating resistors during the first 10 ms of the 20 ms of the simulation. The resulting heat source generated is about $2 \cdot 10^{13}$ W/m^3, corresponding to 100 mW (see Figure 15-18).

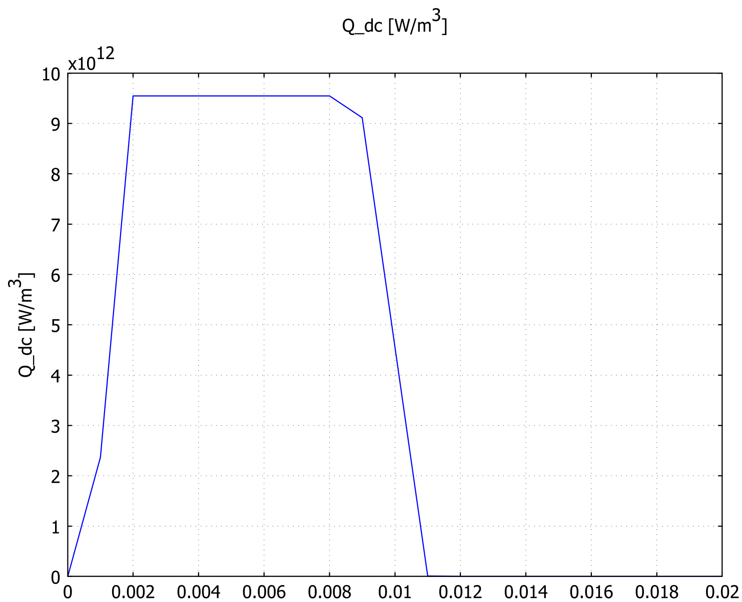


Figure 15-18: Heat source vs. time in the heating resistors.

An important part of the simulation is the modeling of the cooling effects at the boundaries. Experimentally, it has been verified that most of the heat is dissipated in the silicon structure, that is, the "body" of the robot. Also, a minor cooling effect at the tip of the leg has been observed. Motivated by this, the model uses a heat transfer coefficient of $1 \cdot 10^6$ W/(m^2·K) on the part of the leg connected to the rest of the robot, and a coefficient of $1 \cdot 10^5$ W/(m^2·K) on the tip of the leg. These values have been chosen somewhat arbitrarily, but the results have been calibrated with experimental data.

The boundary conditions for the structural mechanics part are simply constrained displacement at the left end of the robot leg and zero force on the rest of the structure.

The 3D model does not include the thin layer of Al and SiO. On the resulting boundary, a structural shell application mode and two specific thermal boundary conditions model the highly conductive and poorly conductive layer.

The heat propagates by conduction through the leg, and thermal expansion and different thermal expansion coefficients of the several materials induce a bending action.

Results and Discussions

Figure 15-19 below shows the bending action at t = 0.2 ms. The plot also includes the temperature at the boundary.

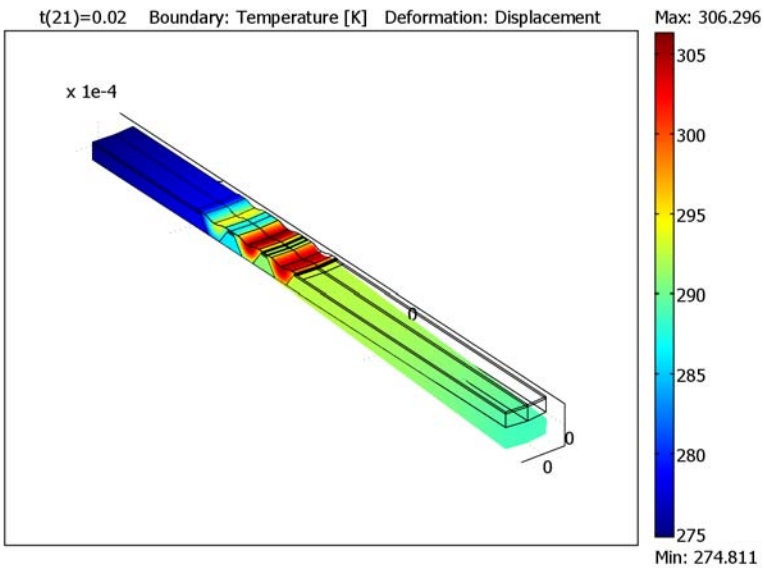


Figure 15-19: Temperature distribution in the microrobot at 20ms

The plots shows that the leg bends as expected with the increase of temperature and moves toward its original position when the heat source is turned off.

Figure 15-20 below shows the total displacement as a function of time at the tip of the leg.

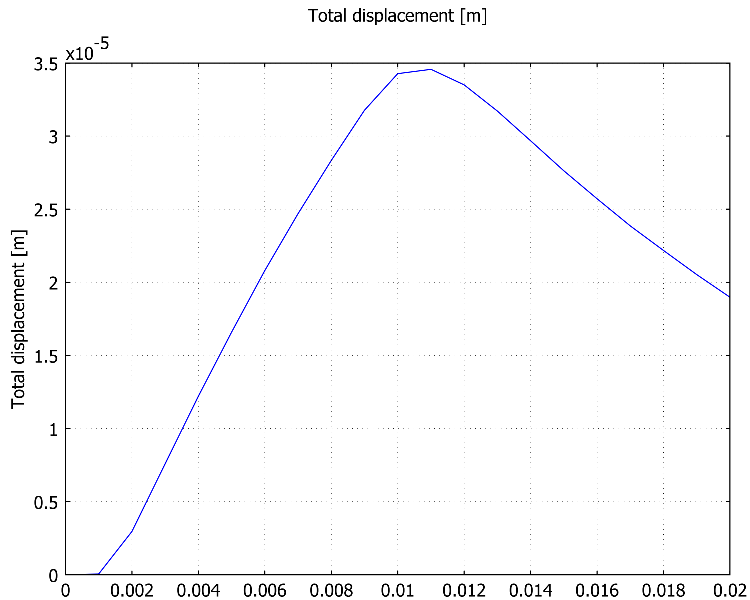


Figure 15-20: Total displacement at the tip of the leg

To verify the validity of the highly conductive and resistive layer, a cross-section plot of the temperature focuses on the middle of the V-groove joints (see Figure 15-21).

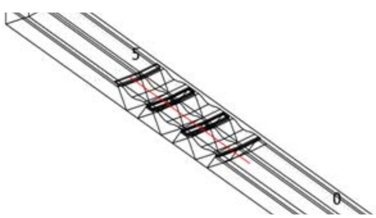


Figure 15-21: Position of the cross-section plot.

You can observe the effect of the thin layer at the positions $z = -1.1 \cdot 10^{-4}$, $-0.6 \cdot 10^{-4}$, $-0.2 \cdot 10^{-4}$, $0.2 \cdot 10^{-4}$, $0.6 \cdot 10^{-4}$, and $1.1 \cdot 10^{-4}$ m in a cross-section plot of the temperature inside the microrobot leg (see Figure 15-22).

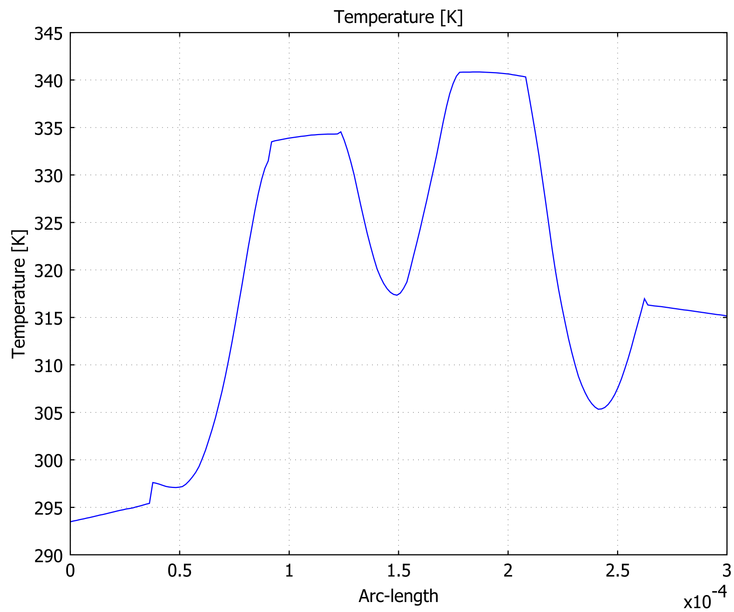


Figure 15-22: A cross-section plot of the temperature distribution inside the microbot leg.

The small dips at $z = -0.2 \cdot 10^{-4}$ m and $1.1 \cdot 10^{-4}$ m correspond well with a 2D model where the thin composite layers are modeled using solids instead of the highly and resistive conductive layers and structural shell elements.

Modeling in COMSOL Multiphysics

The original 2D model consists of thin layers, and it is essential to remove these layer in order to reduce the mesh size in the 3D model. Instead, apply the following boundary conditions:

- For the thermal part, a highly conductive boundary condition replaces the highly conductive layer of aluminum (see Figure 15-16):

$$-n \cdot (-k\nabla T) = -d\rho C\frac{\partial T}{\partial t} - \nabla \cdot (-dk\nabla T)$$

- Represent the resistive layer (see Figure 15-16) with the stiff spring condition

$$-n \cdot (-k\nabla T_2) = \frac{\mathrm{klayer}}{d} \cdot (T_1 - T_2)$$

where klayer is the thermal conductivity of the material, d the thickness of the layer, and T_1 and T_2 are the temperatures on the each side of the boundary.

Note that because the stiff spring boundary condition models a temperature discontinuity on the boundary, you need two General Heat Transfer applications modes.

- The structural part includes a shell element with thermal expansion to model the two thin layers. The software handles the coupling between the shell and the solid elements automatically.

Figure 15-23 shows where the shell application and the thermal boundary conditions are active.

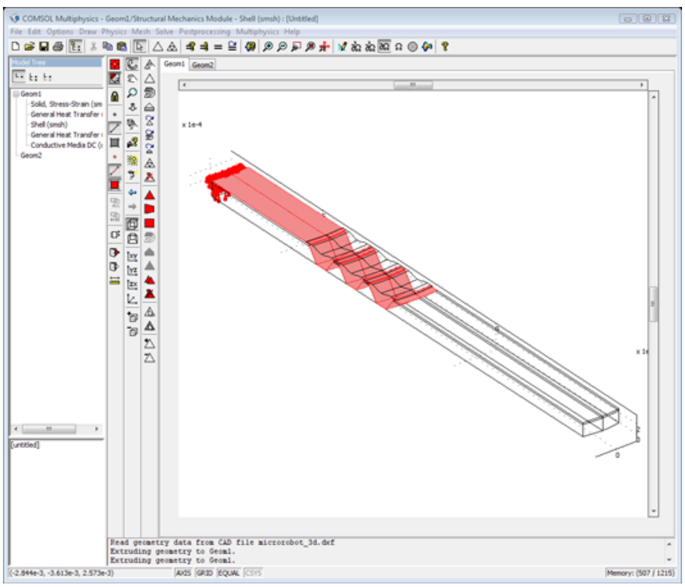


Figure 15-23: Location of the shell element and thermal boundary condition.

Because the time scale of the structural mechanics part is much smaller compared with the heat transfer part, you can neglect the mass effects in the structural analysis. Instead use a parametric analysis to solve the structural mechanics part. This problem only couples the solid analysis to the heat transfer analysis, and not the other way around, so you can use sequential solution procedure: First solve the heat transfer and conductive media DC equations, and then, after storing the solution in a file and setting the linearization point to the stored solution, solve the structural part using the parametric analysis with the time variable t as the parameter.

Reference

1. T. Ebefors, "*Polyimide V-groove Joints for Three-Dimensional Silicon Transducers,*" Ph.D. thesis, May 2000, Dept. of Signal, Sensors and Systems (S3), Royal Institute of Technology (KTH), Stockholm, Sweden.

Model Library path: `Structural_Mechanics_Module/Thermal-Structural_Interaction/microrobot_3d`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Start COMSOL Multiphysics or click the **New** button to open the **Model Navigator**.
2. Select **3D** in the **Space dimension** list.
3. Click the **Multiphysics** button.
4. Select **Structural Mechanics Module>Thermal-Structural Interaction> Solid, Stress-Strain with Thermal Expansion** and then **Quasi-static analysis**.
5. Click **Add**.
6. Open **Structural Mechanics Module>Shell** and select **Quasi-static analysis**.
7. Click **Add**.
8. Open **Heat Transfer Module>General Heat Transfer** and select **Transient Analysis**.
9. Click **Add**.
10. Open **COMSOL Multiphysics>Electromagnetics>Conductive Media DC**.
11. Click **Add**.
12. Click **OK**.

GEOMETRY MODELING

You obtain the geometry of this model by extruding the mesh of the 2D cross-section geometry. The 2D geometry is provided as a DXF file.

1. Go to the **Draw** menu, and select **Work-Plane Settings**.

2 Click the **y-z plane** button, then click **OK**.

3 On the **File** menu point to **Import**, then click **CAD Data From File**.

4 Browse to `COMSOL_Multiphysics/Multiphysics` in the COMSOL models folder.

5 Select `microrobot.dxf`, then click **OK**.

6 Click the **Zoom Extents** button.

7 Click the **Coerce to Solid** button.

8 On the **Draw** menu choose **Extrude**, then define the following extrusion parameters: **Distance** as `4e-5`, **Scale y** as `0.95`, and **Displacement x** as `5e-6`.

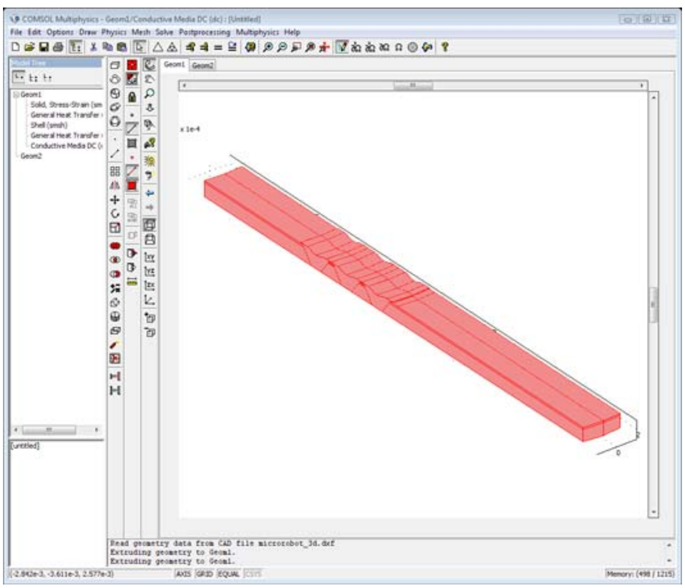
9 Click **OK**.

10 Click the **Geom2** tab.

11 Select **Extrude** from the **Draw** menu and use the same settings except for the **Distance**. In the **Distance** edit field type `-4e-5`.

12 Click **OK**.

13 Select both EXT1 and EXT2, then click the **Coerce to Solid** button on the Draw toolbar.



PHYSICS SETTINGS

Because the microrobot legs consists of several materials with different material properties, you have to define subdomain settings for each materials. Using subdomain expression variables makes it easier to enter the subdomain properties of the model.

Define the following parameters for the materials:

- The thermal conductivity, `k`
- The heat capacity, `C`
- The density, `rho`
- Young's modulus, `E`
- Poisson's ratio, `nu`
- The thermal expansion coefficient, `alpha`

Also define a new variable, `Temperature`, to visualize the temperature in all domains in the postprocessing analysis.

Subdomain Expressions

1 On the **Options** menu, point to **Expressions** and then click **Subdomain Expressions**.

2 In the **Subdomain Expressions** dialog box, enter the following material properties:

SETTINGS	SUBDOMAINS 1, 4, 7, 10, 12, 15, 18, 21	SUBDOMAINS 2, 5, 8, 11, 13, 16, 19, 22	SUBDOMAINS 3, 14	SUBDOMAINS 6, 9, 17, 20
`k`	`140`	`15`	`0.16`	`20`
`C`	`707`	`1000`	`2000`	`707`
`rho`	`2300`	`2865`	`2700`	`2300`
`E`	`130e9`	`95e9`	`2e9`	`160e9`
`nu`	`0.3`	`0.25`	`0.35`	`0.3`
`alpha`	`2.5e-6`	`0.8e-6`	`140e-6`	`2e-6`
`Temperature`	`T`	`T`	`T2`	`T`

From left to right the material properties represent Si, SiN, P, and pSi, respectively

3 Click **OK**.

Use shell elements to model the thin layer. This means that you must define the material properties as boundary expressions.

4 On the **Options** menu, point to **Expressions** and then click **Boundary Expressions**.

5 Enter the following material properties:

SETTINGS	BOUNDARIES 8, 11, 14, 26, 27, 31, 34, 36, 48, 49, 53, 56, 58, 70, 71, 73, 83, 86, 89, 101, 102, 106, 109, 111, 123, 124, 128, 131, 133, 145, 146, 148	BOUNDARIES 15, 17, 20, 37, 39, 42, 59, 61, 64, 90, 92, 95, 112, 114, 117, 134, 136, 139
`E_layer`	`70e9`	`70e9`
`nu_layer`	`0.3`	`0.26`
`alpha_layer`	`23e-6`	`2.35e-5`
`thickness`	`2e-6`	`2e-6`
`rho_layer`	`1410`	`1805`

Subdomain Settings

1 On the **Multiphysics** menu, select **Solid, Stress-Strain(smsld)**.

2 Go to **Physics>Subdomain Settings** and select all subdomains.

3 Enter the following settings for the material properties on the **Material** page:

PROPERTY	VALUE
E	`E`
ν	`nu`
α	`alpha`
ρ	`rho`

4 Click the **Load** tab, and select the **Include thermal expansion** check box.

5 Select Subdomains 3 and 14, then type `T2` in the **Temp** edit field.

6 Select all subdomains and type `273.15` in the **Tempref** edit field.

7 Click **OK**.

8 In the **Multiphysics** menu, select **General Heat Transfer (htgh)**.

9 Go to the **Subdomain Settings** dialog box and select all the subdomains.

10 Enter the following material properties: **Thermal conductivity** `k`, **Density** `rho` and **Heat capacity** `C`.

11 Select Subdomains 6, 9, 17, and 20, then type `Q_dc` in the **Heat source** edit field.

The coupling between the electric and heat transfer problems is now complete. The resistive heating from the Current Media DC application mode is used as the heat source in the General Heat Transfer application to evaluate the resulting temperature distribution.

1 Select Subdomains 3 and 14, then clear the **Active in this domain** check box.

2 Click **OK**.

3 Select **General Heat Transfer (htgh2)** in the **Multiphysics** menu.

4 In the **Subdomain Settings** dialog box, select all subdomains except 3 and 14, then clear the **Active in this domain** check box.

5 Select Subdomains 3 and 14, then set the **Thermal conductivity** to `k`, the **Heat capacity** to `C`, and the **Density** to `rho`.

6 Click **OK**.

7 Select **Conductive Media DC (dc)** in the **Multiphysics** menu.

8 In the **Subdomain Settings** dialog box, select Subdomains 6, 9, 17, and 20.

9 Type `1e8` in the **Electric conductivity** edit field.

10 Select all other subdomains and clear the **Active in this domain** check box.

11 Click **OK**.

Boundary Conditions

1 Select the **Solid, Stress-strain** application mode.

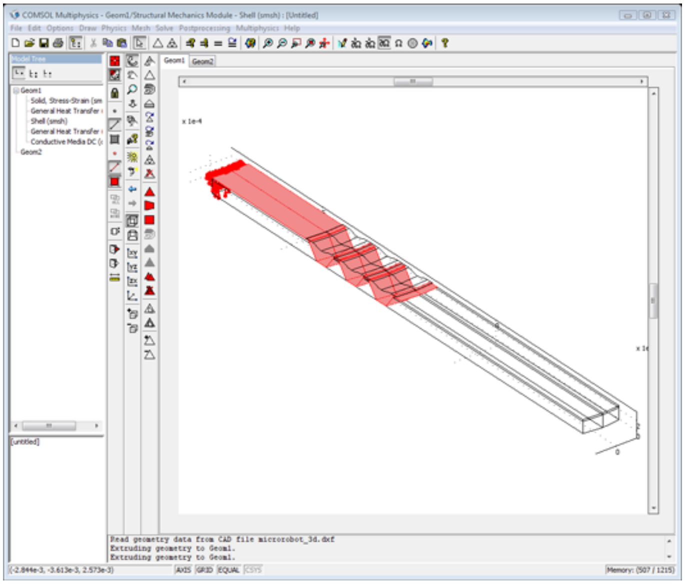
2 Select **Physics>Boundary Settings**, and select **Fixed** from the **Constraint condition** list on the **Constraint** page for Boundaries 74, 75, 149, and 150.

3 Click **OK**.

4 Select **Shell (smsh)** in the **Multiphysics** menu.

5 In the **Boundary Settings** dialog box, select Boundaries 8, 11, 14, 15, 17, 20, 26, 27, 31, 34, 36, 37, 39, 42, 48, 49, 53, 56, 58, 59, 61, 64, 70, 71, 73, 83, 86, 89, 90,

92, 95, 101 102, 106, 109, 111, 112, 114, 117, 123, 124, 128, 131, 133, 134, 136, 139, 145, 146, and 148.



6 Set **Young's modulus** to `E_layer`, **Poisson's ratio** to `nu_layer`, and the **thickness** to `thickness`.
7 Select the other boundaries and clear the **Active in this domain** check box.
8 Click the **Load** tab.
9 Select all the active boundaries and then select the **Include thermal expansion** check box.
10 Select Boundaries 11, 14, 15, 20, 26, 27, 31, 34, 36, 37, 42, 48, 49, 53, 56, 58, 59, 64, 70, 71, 86, 89, 90, 95, 101, 102, 106, 109, 111, 112, 117, 123, 124, 128, 131, 133, 134, 139, 145, and 146.
11 In the **Temp** edit field type `T2`, in the **Tempref** edit field type `273.15`, and in the **dT** edit field type `T-T2`.
12 Select Boundaries 8, 73, 83, and 148, then enter `T` as **Temp** and `273.15` as **Tempref**.
13 Select Boundaries 17, 39, 61, 92, 114, and 136, then set `T2` as **Temp** and `273.15` as **Tempref**.
14 Click **OK**.
15 In the **Multiphysics** menu, select **General Heat Transfer (htgh)**.

16 In the **Physics** menu, select **Boundary Settings**.

17 Select Boundaries 74, 75, 149, and 150, then set a **Heat flux** boundary condition with **h** as `1e6`.

18 Select Boundaries 8, 73, 83, and 148. Click the **Highly Conductive Layer** tab, and select the **Enable heat transfer in highly conductive layer** check box.

19 Set $\mathbf{k_s}$ as `238`, ρ_s as `1410`, $\mathbf{C_{p,s}}$ as `900`, and $\mathbf{d_s}$ as `2e-6`.

20 Select Boundaries 11, 14, 26, 27, 31, 34, 36, 48, 49, 53, 56, 58, 70, 71, 86, 89, 101, 102, 106, 109, 111, 123, 124, 128, 131, 133, 145, and 146, then set a **Temperature** boundary condition with $\mathbf{T_0}$ as `T2`.

21 Select Boundaries 15, 20, 37, 42, 59, 64, 90, 95, 112, 117, 134, and 139.

22 Set a **Heat flux** boundary condition with a heat transfer coefficient **h** as `1.4/1e-6` and an external temperature $\mathbf{T_{inf}}$ as `T2`.

23 Select Boundaries 2, 5, 77, and 80.

24 Set a **Heat flux** boundary condition and enter `1e5` in the **h** field.

25 Click **OK**.

26 In the **Multiphysics** menu, select **General Heat Transfer (htgh2)**.

27 In the **Boundary Settings** dialog box, select Boundaries 15, 20, 37, 42, 59, 64, 90, 95, 112, 117, 134, and 139.

28 Set a **Heat flux** condition with **h** as `1.4/1e-6` and $\mathbf{T_{inf}}$ as `T`.

29 Click the **Highly Conductive Layer** tab and select the **Enable heat transfer in highly conductive layer** check box. In the $\mathbf{k_s}$ edit field type `238`, in the ρ_s edit field type `1410`, in the $\mathbf{C_{p,s}}$ edit field type `900`, and finally in the $\mathbf{d_s}$ edit field type `1e-6`.

30 Select Boundaries 17, 39, 61, 92, 114, and 136.

31 Set the **Highly conductive layer** condition with $\mathbf{k_s}$ as `238`, ρ_s as `1410`, $\mathbf{C_{p,s}}$ as `900`, and $\mathbf{d_s}$ as `2e-6`.

32 Click **OK**.

33 In the **Multiphysics** menu, select **Conductive Media DC (dc)**.

34 In the **Boundary Settings** dialog box, select all boundaries and select the **Electric Insulation** boundary condition.

35 Select Boundaries 28 and 50 and set an **Electric potential** condition, with $\mathbf{V_0}$ as `(flc1hs(t[1/s]-1e-3,5e-4)-flc1hs(t[1/s]-1e-2,5e-4))*3e-2`.

The electric potential is set to 30 mV during 9 ms starting at t = 1 ms. The smoothed Heaviside function `flc1hs` ensures continuity in the potential condition.

36 Select Boundaries 156 and 159, then set a **Ground** boundary condition.

37 Click **OK**.

MESH

1 From the **Mesh** menu choose **Free Mesh Parameters**.

2 From the **Predefined mesh sizes** list select **Coarser**.

3 Click **OK**.

COMPUTING THE SOLUTION

1 From the **Solve** menu choose **Solver Manager**.

2 Click the **Solve For** tab.

3 In the **Solve for variables** list select only the application modes **General Heat Transfer (htgh)**, **General Heat Transfer (htgh2)**, and **Conductive Media DC (dc)** (or, equivalently, the variables **V**, **T**, and **T2**).

4 Open the **Solver Parameters** dialog box by clicking on the corresponding toolbar button.

5 On the **General** page, type `0:0.001:0.02` in the **Times** edit field. This provides the solution at 21 equally-spaced time steps from 0 to 20 ms or every 1 ms. The tolerances for the time-dependent solver control the actual time stepping.

6 Click **OK**.

7 Click the **Solve** button on the Main toolbar.

8 Go back to the **Solver Manager**, select the structural mechanics application modes on the **Solve For** page.

9 On the **Initial Value** page click the **Initial value expression** option button.

10 Click the **Store Solution** button, then click **OK** in the **Store Solution** dialog box to store all solutions.

11 In the **Values of variables not solved for and linearization point** area click the **Stored solution** option button and select **All** in the **Solution at time** list.

12 Click **OK**.

13 In the **Solver Parameters** dialog box select the **Parametric** solver.

14 In the **Parameter name** edit field type `t` and in the **Parameter values** edit field type `0:0.001:0.02`.

15 Click **OK**.

16 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 From the **Postprocessing** menu open the **Plot Parameters** dialog box.

2 In the **Plot type** area on the **General** page clear the **Slice** check box and select the **Boundary** check box.

3 Go to the **Boundary** page and type `Temperature` in the **Expression** edit field.

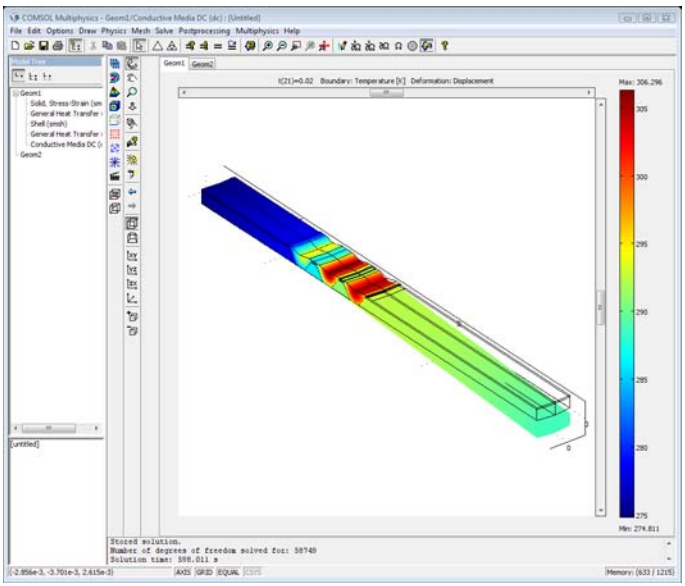
4 Select the **Deformed Shape** check box to plot using a deformed shape.

5 Click the **Deform** tab.

6 In the **Deformation data** area, select **Solid, Stress-Strain (smsld)>Displacement** in the **Predefined quantities** list on the **Subdomain Data** tab.

7 To get the following plot, clear the **Auto** check box for automatic scaling of the deformed shape. Then type `1` in the **Scale factor** edit field.

8 Click **OK**.



9 To get an animation of the results, click the **Animate** button on the Plot toolbar.

Thermal Expansion in a MEMS Device Using the Material Library

The purpose of this model is to exemplify the use of the Material Library in COMSOL Multiphysics. This library contains more than 20,000 property functions for 2500 materials. The larger part of these properties are mechanical and thermal properties for solid materials given as functions of temperature. You need the Material Library to build the model.

Introduction

Thermal expansion is a common method used in the microscale to displace a part of a component, for example in an actuator. In this example model the opposite is required; that is, there should be a minimum of thermal expansion. Such a device could be included in a microgyroscope or any other sensor for acceleration or positioning.

Model Definition

Figure 15-24 below shows the model geometry:

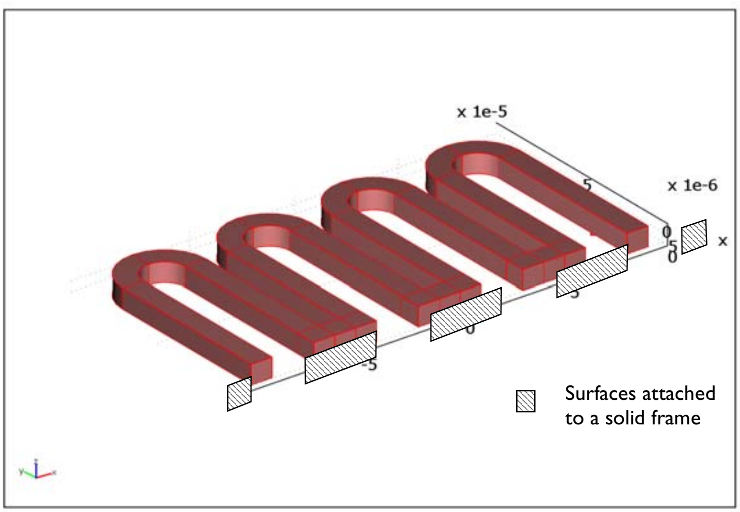


Figure 15-24: Geometry of the device.

The model consists of two sets of physics:

- A thermal balance with a heat source in the device, originating from Joule heating (ohmic heating). Air cooling is applied on the boundaries except at the position where the device is attached to a solid frame, where an insulation condition is set.
- A force balance for the structural analysis with a volume load caused by thermal expansions. The device is fixed at the positions where it is attached to a solid frame (see Figure 15-24).

The device is made of the copper-beryllium alloy UNS C17500.

The thermal balance consists of a balance of flux at steady state. The heat flux is given by conduction only. The heat source is a constant heat source of $1 \cdot 10^8$ W/m^3. The air cooling at the boundaries is expressed using a constant heat transfer coefficient of 10 W/m^2 and an ambient temperature of 298 K.

The stress and strains are well within the elastic region for the material. The expression for thermal expansion requires a strain reference temperature for the copper-beryllium alloy, which in this case is 293 K.

All other thermal and mechanical properties are obtained from the Material Library.

Results and Discussion

The following figure shows the temperature distribution in the device. The heat source increases the temperature to 323 K from an ambient temperature of 298 K. The temperature varies less than 1/100 of a degree in the device. The displacements vary accordingly, and the model shows that it is possible to study the device using only one unit cell, for example, 1/2 of a U-shaped section.

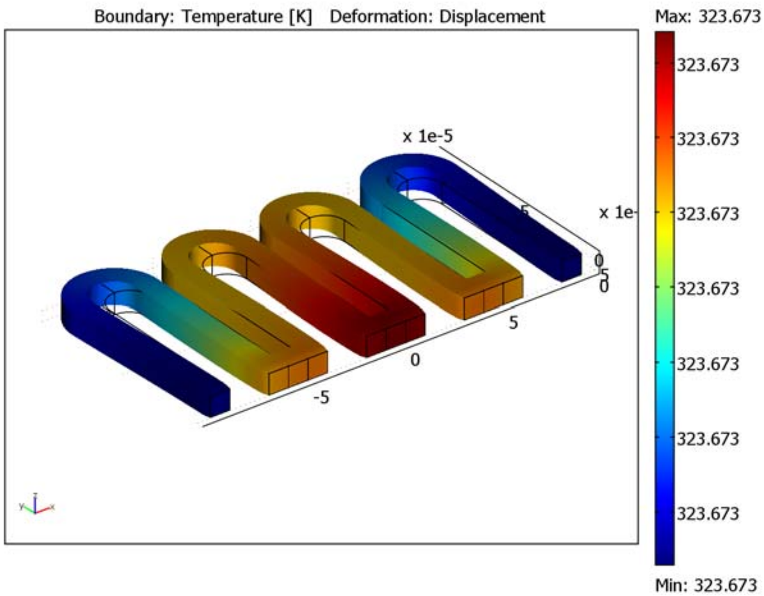


Figure 15-25: Temperature and displacement of the device. The edges of the original geometry are shown in black. The deformed shape is exaggerated by a factor of almost 200.

The following figure (Figure 15-26) shows the displacement of a curve that follows the top inner edges of the device from left to right. It is clear from Figure 15-26 that the displacement follows a repetitive pattern along the length of the device. This also supports the hypothesis that 1/2 of a U-shaped section is enough to represent the behavior of the device.

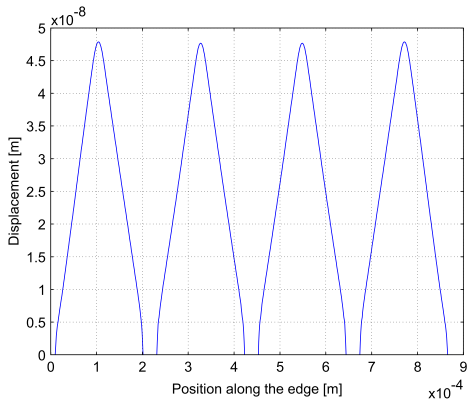


Figure 15-26: The displacement along the inner edge of the device shows a repetitive pattern, which means that a model of half of the u-shaped structure is enough to study the effects of thermal expansion on the device.

Model Library path: `Structural_Mechanics_Module/Thermal-Structure_Interaction/thermal_expansion`

Modeling Using the Graphical User Interface

1 Open the **Model Navigator**.

2 Select **3D** from the **Space dimension** list.

3 Select **Structural Mechanics Module>Thermal-Structural Interactions>Solid, Stress-Strain with Thermal Expansion>Static analysis**.

4 Click **OK**.

GEOMETRY MODELING

1 From the **Draw** menu, select **Work Plane Settings**.

2 Click the **x-y** plane option button if it is not already selected.

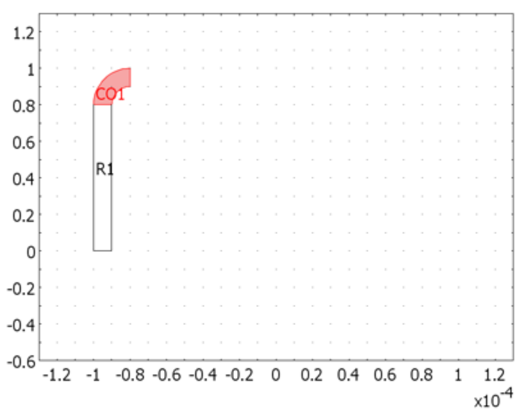
3 Click **OK**.

4 From the **Options** menu, select **Axis/Grid Settings**.

5 Enter `-1.3e-4` in the **x min** edit field, `1.3e-4` in the **x max** edit field, `-0.3e-4` in the **y min** edit field, and `1.3e-4` in the **y max** edit field.
6 Click the **Grid** tab.
7 Clear the **Auto** box.
8 Enter `1e-5` in the **x spacing** and `1e-5` in the **y spacing** edit fields, respectively.
9 Click **OK**.

Next, draw a projection of the 3D geometry and then extrude it to create a 3D object:

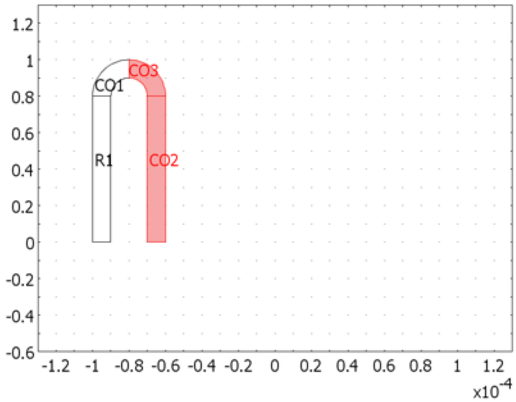
1 From the **Draw** menu, select **Specify Objects>Rectangle**.
2 Type `1e-5` in the **Width** and `8e-5` in the **Height** edit fields, respectively.
3 Type `-1e-4` in the **x** edit field in the **Position** area using the **Corner** option in the **Base** list.
4 Click **OK**.
5 Click the **2nd Degree Bezier Curve** tool button.
6 Click the coordinates (-9e-5, 8e-5), (-9e-5, 9e-5), and (-8e-5, 9e-5) to create an arc.
7 Click the **Line** tool button.
8 Click the coordinate (-8e-5, 1e-4).
9 Click the **2nd Degree Bezier Curve** tool button.
10 Click the coordinates (-1e-4, 1e-4) and (-1e-4, 8e-5).
11 Click the right mouse button to form a composite object with the shape of an elbow.



12 Press Ctrl+A to select all objects.
13 Click the **Mirror** tool button.

14 Enter `-8e-5` in the **x** edit field in the **Point on line** row in the **Paste** dialog box. Make sure that the normal vector to the line has the default values $(1, 0)$.

15 Click **OK**.



16 Press Ctrl+A to select and Ctrl+C to copy all objects.

17 Press Ctrl+V to paste all objects.

18 Enter `5e-5` in the **x** edit field for **Displacements** in the **Paste** dialog box.

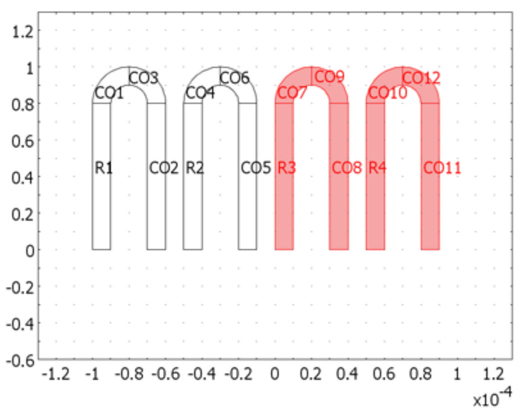
19 Click **OK**.

20 Press Ctrl+A to select and Ctrl+C to copy all objects.

21 Press Ctrl+C to paste all objects.

22 Enter `1e-4` in the **x** edit field for **Displacements** in the **Paste** dialog box.

23 Click **OK**.



24 Click **Zoom Extents**.

25 From the **Draw** menu, select **Specify Objects>Rectangle**.

26 Enter `3e-5` in the **Width**, `1e-5` in the **Height**, and `-7e-5` in the **x** edit fields, respectively.

27 Click **OK**.

28 Press Ctrl+C followed by Ctrl+V to copy and paste the selected rectangle.

29 Type `5e-5` in the **x** edit field for **Displacements**.

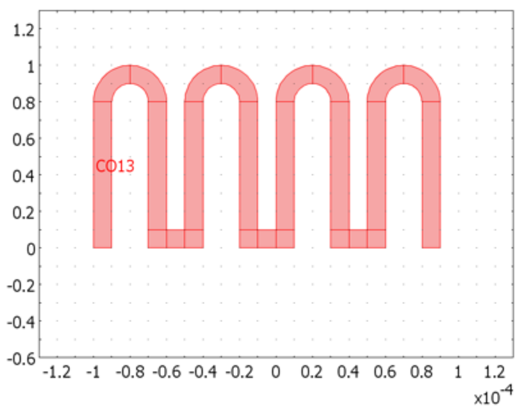
30 Click **OK**.

31 Press Ctrl+C followed by Ctrl+V to copy and paste the selected rectangle.

32 Type `5e-5` in the **x** edit field for **Displacements**.

33 Click **OK**.

34 Press Ctrl+A and click the **Union** button on the Draw toolbar to create one composite object.



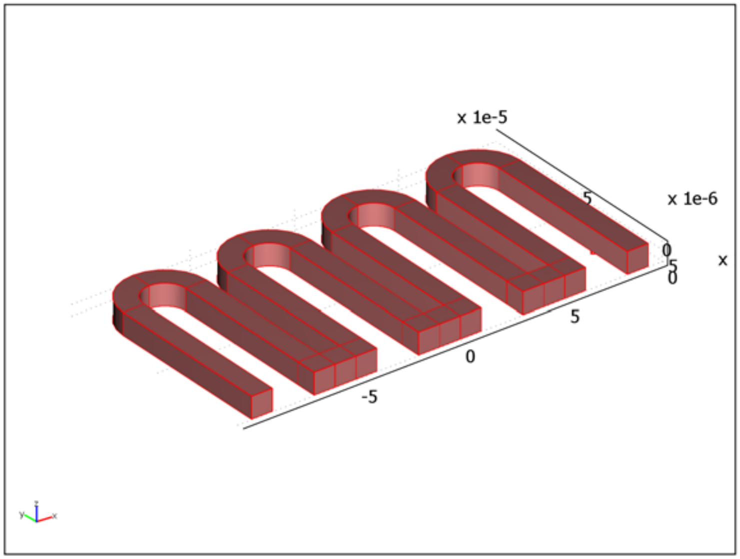
35 From the **Draw** menu, select **Extrude**.

36 Enter `1e-5` in the **Distance** edit field.

37 Click **OK**.

38 Click the **Headlight** button on the Camera toolbar.

The completed geometry in the drawing area should now look like that in the following figure.



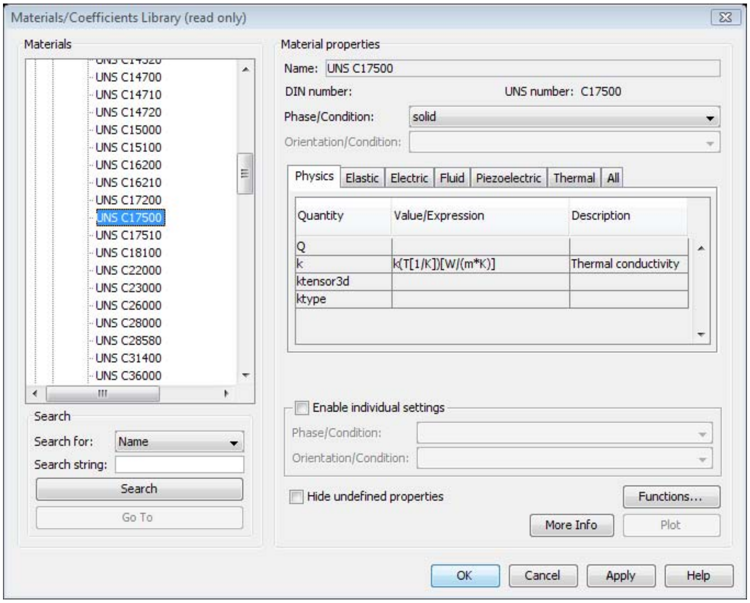
PHYSICS SETTINGS

Start by setting the properties and boundary conditions for the thermal analysis.

Subdomain Settings—Heat Transfer by Conduction

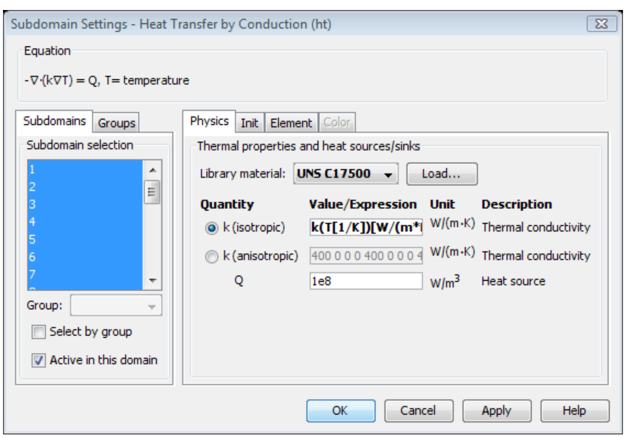
1 From the **Physics** menu, select **Subdomain Settings**.
2 Click the **Subdomain selection** list and press Ctrl+A to select all subdomains.
3 Click the **Load** button.
4 Click the **Material Library** node, which is second in the **Materials** list.
5 Click the **Al & Cu alloys** node.
6 Click the **Copper alloys** node.

7 Select the copper alloy **UNS C17500** from the list.



8 Click **OK**.

9 Enter 1e8 in the **Heat source** edit field.



10 Click **OK**.

Boundary Conditions—Heat Transfer by Conduction

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

2 Click the **Boundary selection** list.

3 Press Ctrl+A to select all boundaries.

4 In the **Boundary selection** list, press Ctrl and click Boundaries 2, 17, 26, 32, 52, 61, 67, 87, 96, 102, and 122 to clear them from the current selection. Note that you can also do this directly in the user interface by selecting all boundaries, clicking the right mouse button to keep the selection, and then selecting the boundaries corresponding to the attachment to the base one at the time. When you click on a boundary, it turns green. If this is the boundary you want to clear from the selection list, click the right mouse button. Repeat this procedure until all boundaries listed above are cleared from the selection.

5 Select **Heat Flux** from the **Boundary condition** list.

6 Enter `10` in the **Heat transfer coefficient** edit field and `298` in the **External temperature** edit field.

7 Click **OK**.

Subdomain Settings—Solid, Stress-Strain

1 From the **Multiphysics** menu, select the **Solid, Stress-Strain** application mode.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select all the subdomains in the **Subdomain selection** list, if they are not already selected.

4 In the **Library material** list, select **UNS C17500**.

5 Click the **Load** tab. Note that the **Include thermal expansion** box is selected and that the variable for the temperature field, `T`, already appears in the **Strain temperature** edit field.

6 Type `293` in the **Strain ref. temperature** edit field. You can get the strain reference temperature from the Material Library. In this case, it is included in the notes for the thermal expansion coefficient function, alpha, in the Material Library.

7 Click **OK**.

Boundary Settings—Solid, Stress-Strain

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

2 Click the **Boundary selection** list and press Ctrl+D to clear the current selection.

3 Press Ctrl and click Boundaries 2, 17, 26, 32, 52, 61, 67, 87, 96, 102, and 122 to select the surfaces attached to the solid frame.

4 Select **Fixed** from the **Constraint condition** list.

5 Click **OK**.

MESH GENERATION

1 Click the **Interactive Mesh** button.

2 Select Boundary 2, which is the left-most boundary attached to the solid frame.

3 Click the **Decrease Mesh Size** button to decrease the mesh size from normal to fine.

4 Click the **Decrease Mesh Size** button a second and a third time to decrease the mesh size to extra fine.

5 Click the **Mesh Selected (Mapped)** mesh button.

6 Click the **Subdomain Mode** button.

7 Select Subdomains 1–5, which are the five first domains from left to right. Note that you can do this by clicking the mouse until you get the desired subdomain highlighted, then you can right-click to save the selection. Then continue to the next subdomain, highlight it by clicking on it, and right-click to save the selection and so on until you have selected all five subdomains.

8 Click the **Mesh Selected (Swept)** button.

   The reason for not sweeping the mesh throughout all subdomains is that the meshing of the cube sections, which unite the U-shaped sections of the geometry, requires that you change the sweep direction in order to create only hexagonal mesh elements. If you instead mesh all U-shaped sections first, COMSOL Multiphysics detects the sweep dimension automatically by the fact that two opposite faces of the cubes are meshed. You can therefore first create a swept mesh for all U-shaped sections and then mesh the cubes to create a hexagonal mesh for the whole geometry.

9 Click the **Boundary Mode** button.

10 Select Boundaries 32, 67, and 102 by clicking on them in the user interface. Note that you can use the same strategy as outlined above—that is, to click on the desired boundary until it gets highlighted—then right-click to save the selection, and so on for multiple selections.

11 Click the **Mesh Selected (Mapped)** button.

12 Click the **Subdomain Mode** button.

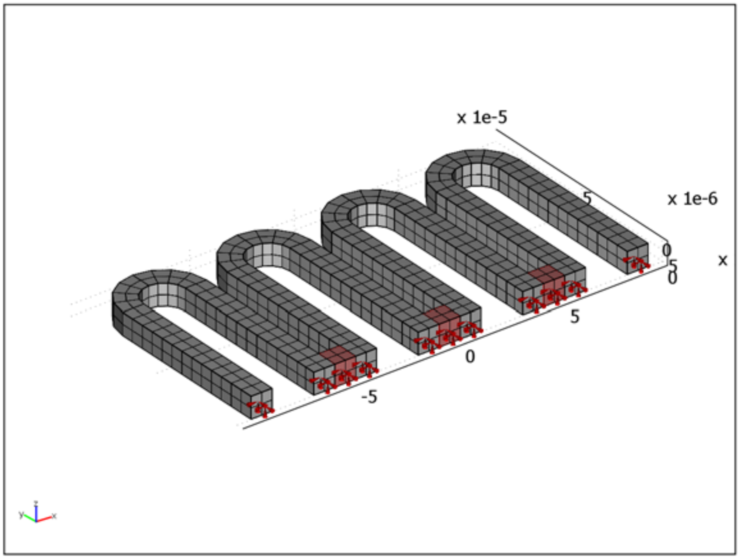
13 Press Ctrl+D to clear the current selection.

14 Select Subdomains 7–12, 14–19, and 21–25 by using the strategy outlined in Point 7 above.

15 Click the **Mesh Selected (Swept)** button.

16 Select Subdomains 6, 13, and 20.

17 Click the **Mesh Selected (Swept)** button.



COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar. Note that the solution is fully coupled, which is not required in this case. You can solve the thermal balance first because it does not depend on the structural analysis. The structural analysis does require the temperature field in order to calculate the loads due to thermal expansion. If you are familiar with the use of the Solver Manager, you can use it to solve the problem sequentially.

POSTPROCESSING AND VISUALIZATION

To generate the plot in Figure 15-25, visualizing the surface temperature and the structural deformation of the device, follow these instructions:

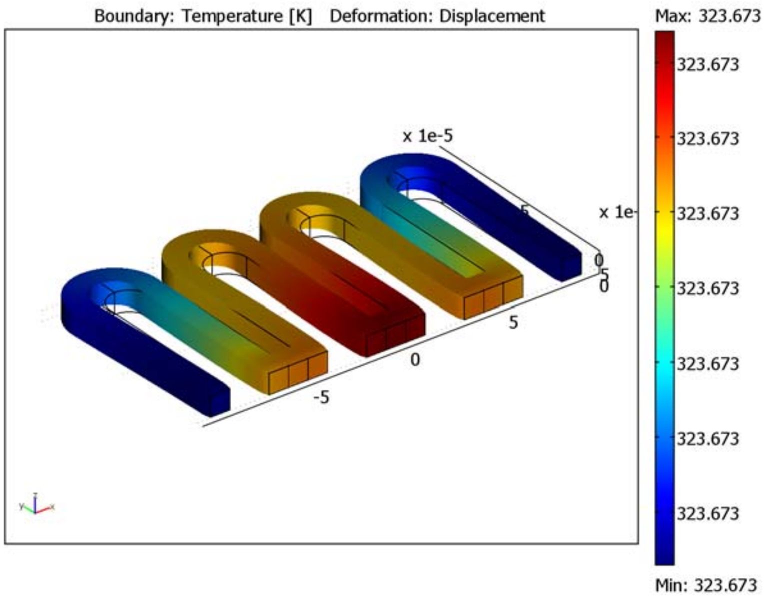
1 Click the **Plot Parameters** button.
2 Click the **General** tab.
3 Clear the **Slice** box.
4 Select the **Boundary** box.
5 Select the **Deformed shape** box.
6 Click the **Boundary** tab.

7 In the **Expression** edit field type T.

8 Click the **Deform** tab.

9 On the **Subdomain Data** page, select **Solid, Stress-Strain (smsld)>Displacement** in the **Predefined quantities** list (the default option).

10 Click **OK**.



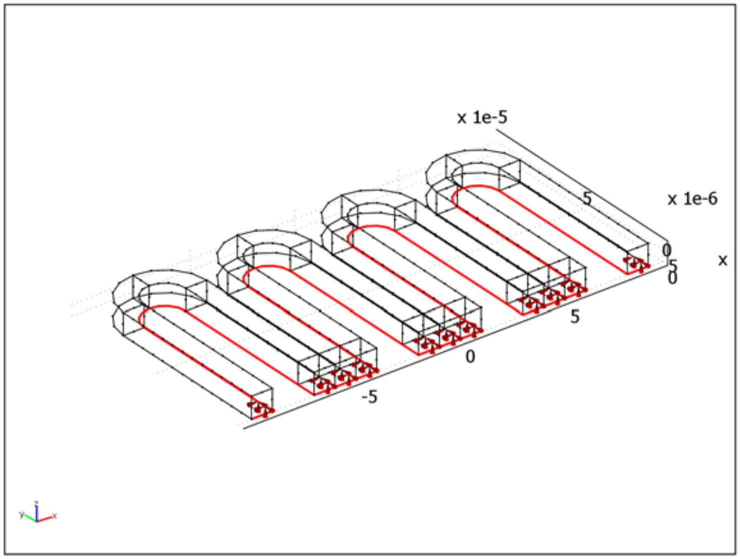
Create the deformation plot in Figure 15-26 as follows:

1 Choose **Postprocessing>Cross-Section Plot Parameters**.

2 Click the **Line/Extrusion** tab.

3 Select the inner boundary segments marked in the following figure. To add a segment that you have highlighted in red by clicking with the mouse to the selection

use right-click. The segment should then turn blue in the drawing area. Repeat this procedure until you have selected all inner boundary segments.



4 In the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>Total displacement**.

5 On the **General** page, click the **Title/Axis** button.

6 Click the option button next to the **Title** edit field. Leave the edit field empty to obtain a plot without title.

7 Click the option button next to the **First axis label** edit field, then enter the label `Position along the edge [m]`.

8 Click the option button next to the **Second axis label** edit field, then enter the label `Displacement [m]`.

9 Click **OK** to close the **Title/Axis Settings** dialog box.

10 Click **OK** to close the **Cross-Section Plot Parameters** dialog box and generate the plot.

Heat Generation in a Vibrating Structure

Introduction

When a structure is subjected to vibrations of high frequency, a significant amount of heat can be generated within the structure because of mechanical (viscoelastic) losses in the material. A second mechanism contributing to the slow temperature rise in a vibrating structure is called thermoelastic damping and represents the energy transfer between the thermal and mechanical domains.

In this example, you compute a fully coupled thermoelastic response for a vibrating beam-like structure by combining the stress-strain analysis with the linearized heat-transport equation. The analysis is performed in the frequency domain.

You then model the slow rise of the temperature in the beam as a transient heat-transfer problem. The corresponding heat-transfer equation contains two source terms computed using the results of the frequency response analysis. These terms represent the heat generation due to mechanical losses in the material and nonlinear effects related to the thermoelastic damping.

Model Definition

The beam consists of two layers of different materials. One end of the beam is fixed, and the other one is subjected to periodic loading that is represented in the frequency domain as $F_z \exp(j\omega t)$, where j is the imaginary unit, and ω is the angular frequency.

The starting point of the thermal analysis is the heat-transfer equation

$$T\frac{\partial S}{\partial t} - \nabla \cdot (k\nabla T) = Q \qquad (15\text{-}2)$$

where k is the thermal conductivity matrix. For a linear thermo-elastic solid, the entropy per unit volume is

$$S = \rho C_p \log(T/T_0) + S_{\mathrm{elast}} \qquad (15\text{-}3)$$

where, in accordance with the Dulong-Petit law, the volumetric heat capacity ρC_p is independent of the temperature. Furthermore,

$$S_{\text{elast}} = \alpha_{\text{vec}} \cdot D[\varepsilon - \alpha_{\text{vec}}(T - T_0)] \tag{15-4}$$

where D is the elasticity matrix, ε is the strain vector, α_{vec} is the thermal expansion vector, and T_0 is the strain reference temperature.

The initial state at time $t = 0$ is stress-free, and the initial temperature is T_0 across the beam.

The temperature field is decomposed into small-amplitude periodic oscillations and slow temperature rise. The total (viscoelastic) stress that appears in the frequency response analysis is given by

$$\sigma = D[(1 + j\eta)\varepsilon - \alpha_{\text{vec}} T_1] \tag{15-5}$$

where η is the loss factor and T_1 presents the linear temperature response in the frequency domain. The corresponding temperature oscillations in the physical domain are

$$\mathrm{Real}[T_1(x_i)\exp(j\omega t)]. \tag{15-6}$$

The elastic part of the entropy in the frequency domain is

$$S_{\text{elast}} = \alpha_{\text{vec}} \cdot (\sigma - j\eta D\varepsilon) \tag{15-7}$$

Linearizing Equation 15-2 and applying the frequency decomposition gives

$$-\nabla \cdot (k\nabla T_1) = Q_1 \tag{15-8}$$

where

$$Q_1 = -j\omega(\rho C_p T_1 + T_0 S_{\text{elast}}) \tag{15-9}$$

Use Equation 15-8 to compute the temperature response. In the frequency response analysis, use the temperature T_1 as the strain temperature and set the strain reference temperature to zero. This is because the solution physically corresponds to thermoelastic oscillations of small amplitude, which initially are zero. However, T_0 in Equation 15-9 is the actual temperature of the beam at rest.

Using the results of the frequency response analysis, model the slow temperature rise with the following equation:

$$\rho C_p \frac{\partial T}{\partial t} - \nabla \cdot (k\nabla T) = Q_2 + Q_d \tag{15-10}$$

Here T represents the temperature averaged over the time period $2\pi/\omega$, and the heat sources are

$$Q_2 = -\frac{1}{2}\mathrm{Real}[T_1\mathrm{Conj}(j\omega S_{\mathrm{elast}})] \tag{15-11}$$

and

$$Q_d = \frac{1}{2}\omega\eta\mathrm{Real}[\varepsilon \cdot \mathrm{Conj}(D\varepsilon)] \tag{15-12}$$

The term Q_d presents the internal work of the nonelastic (viscous) forces over the period; Q_d is related to the energy transfer between the mechanical and thermal domains due to the nonlinear nature of thermo-elastic coupling. For S_{elast} and Q_d use the predefined variables `Ent` and `Qdamp`, respectively.

The initial condition for Equation 15-10 is

$$T = T_0 \tag{15-13}$$

For this model, assume that $T_0 = 300$ K.

Use the following boundary conditions:

- At the fixed end, use the temperature condition $T = T_0$.
- At the end subjected to forcing, use the thermal insulation condition.
- The boundary between the layers of different material is an internal boundary.
- At all other boundaries, use the heat-flux condition

$$\mathbf{n} \cdot (-k\nabla T) = h(T - T_{\mathrm{e}}) \tag{15-14}$$

where h is the heat transfer coefficient and T_{e} is the external temperature.

For the temperature response, the condition at the fixed end is

$$T_1 = 0 \tag{15-15}$$

Use the following condition instead of Equation 15-14 where applicable:

$$\mathbf{n} \cdot (-k\nabla T_1) = hT_1 \tag{15-16}$$

The beam consists of two layers made of aluminum and titanium, respectively, with the corresponding loss factors 0.001 and 0.005.

For the simulation, apply a periodic loading in the *z* direction of magnitude 1.7 MPa and frequency 7767 Hz at the free end of the beam for 2 seconds, keeping the fixed end and the structure environment at a constant temperature of 300 K during the process.

Results and Discussion

Figure 15-27 displays the temperature distribution at the end of the simulated 2-second process. As the figure shows, the maximum temperature rise in the beam is 0.19 °C.

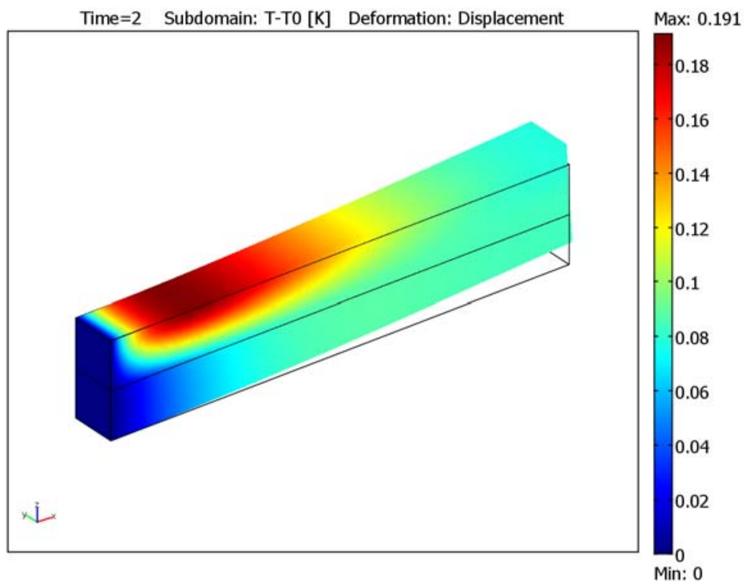


Figure 15-27: Temperature increase in the beam after 2 seconds of forced vibrations.

Reference

1. A. Duwel, R.N. Candler, T.W. Kenny, and M. Varghese, *J. Microelectromech. Sys.*, vol. 15, no. 6, pp. 1437–1445, 2006.

Model Library path: `Structural_Mechanics_Module/Thermal-Structure_Interaction/vibrating_beam`

Modeling Using the Graphical User Interface

1 Start COMSOL Multiphysics.
2 In the **Model Navigator**, select **3D** from the **Space dimension** list, then click the **Multiphysics** button.
3 From the **Application Modes** list, select **Structural Mechanics Module>Solid, Stress-Strain>Frequency response analysis**. Click **Add**.
4 From the **Application Modes** list, select **COMSOL Multiphysics>Heat Transfer>Conduction>Transient analysis**. Click **Add**.
5 Select a second **Conduction** application mode from the same folder, but this time with **Steady-state analysis**. Change the entry in the **Dependent variables** to `T1`, and the **Application mode name** to `ht1`.
6 Click **Add**, then click **OK**.

GEOMETRY MODELING

1 Click the **Block** button on the Draw toolbar.
2 Enter the following values in the corresponding edit fields:

	LENGTH	AXIS BASE POINT	AXIS DIRECTION VECTOR
x	0.01	0	0
y	0.001	0	0
z	0.001	0	1

3 Click **OK** to create block BLK1
4 Repeat Steps 1–3 to create block BLK2 using the following inputs:

	LENGTH	AXIS BASE POINT	AXIS DIRECTION VECTOR
x	0.01	0	0
y	0.001	0	0
z	0.001	0.001	1

MESH GENERATION

1 From the **Mesh** menu, select **Free Mesh Parameters**.

2 On the **Subdomain** page, select both subdomains and enter `4.5e-4` in the **Maximum element size** edit field.

3 Click **Remesh** to generate the mesh.

4 Click **OK**.

OPTIONS AND SETTINGS

Constants

1 From the **Options** menu, open the **Constants** dialog box.

2 Enter the following constants (the descriptions are optional):

NAME	EXPRESSION	DESCRIPTION
eta1	0.001	Loss factor in Subdomain 1
eta2	0.005	Loss factor in Subdomain 2
T0	300[K]	Initial temperature
Te	300[K]	External temperature
he	5[W/(m^2*K)]	Heat-transfer coefficient
Fz	1.7[MPa]	Load amplitude

3 Click **OK**.

Scalar Expressions

1 From the **Options** menu, choose **Expressions>Scalar Expressions**.

2 Enter the following expressions (the descriptions are optional);

NAME	EXPRESSION	DESCRIPTION
Q1	-j*omega_smsld*(rho_ht*C_ht*T1+T0*Ent_smsld)	Heat source for linear temperature response
Q2	-real(0.5/T0*conj(T1)*k_ht1*(T1xx+T1yy+T1zz))	Heat source due to nonlinearity

3 Click **OK**.

PHYSICS SETTINGS

Subdomain Settings—Solid, Stress-Strain

1 From the **Multiphysics** menu, select **1 Solid, Stress-Strain (smsld)**.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Click the **Load** tab.
4 Select both subdomains and select the **Include thermal expansion** check box.
5 Enter `T1` in the **Strain temperature** edit field and `0` in the **Strain ref. temperature** edit field.
6 Click the **Material** tab.
7 Select Subdomain 1, then click the **Load** button.
8 From the **Basic Material Properties** library, select **Aluminum** and click **OK**.
9 Select Subdomain 2, then click the **Load** button.
10 From the **Basic Material Properties** library, select **Titanium beta-21S** and click **OK**.
11 Click the **Damping** tab.
12 From the **Damping model** list, select **Loss factor**. Type `eta2` in the **Loss factor** edit field.
13 Select Subdomain 1.
14 From the **Damping model** list, select **Loss factor**. Type `eta1` in the **Loss factor** edit field.
15 Click **OK**.

Subdomain Settings—Heat Transfer by Conduction

1 From the **Multiphysics** menu, select **2 Heat Transfer by Conduction (ht)**.
2 From the **Physics** menu, select **Subdomain Settings**.
3 Select both Subdomains 1 and 2.
4 On the **Init** page, type `T0` in the **Temperature** edit field.
5 On the **Physics** page, type `Q2+Qdamp_smsld` in the **Heat source** edit field.
6 Select Subdomain 1, then select **Aluminum** from the **Library material** list.
7 Select Subdomain 2, then select **Titanium beta-21S** from the **Library material** list.
8 Click **OK**.

Subdomain Settings—Heat Transfer by Conduction 1

1 From the **Multiphysics** menu, select **3 Heat Transfer by Conduction (ht1)**.
2 From the **Physics** menu, select **Subdomain Settings**.
3 Select both Subdomains 1 and 2.
4 In the **Heat source** edit field, type `Q1`.
5 Select Subdomain 1, then select **Aluminum** from the **Library material** list.
6 Select Subdomain 2, then select **Titanium beta-21S** from the **Library material** list.

7 Click **OK**.

Boundary Conditions—Solid, Stress-Strain

1 From the **Multiphysics** menu select **1 Solid, Stress-Strain (smsld)**.

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

3 Select Boundaries 1 and 4.

4 On the **Constraint** page, set the **Constraint condition** to **Fixed**.

5 Click the **Load** tab. Select Boundaries 10 and 11.

6 In the first part of the **Face load (force/area) z-dir.** edit field, type `Fz`.

7 Click **OK**.

Boundary Conditions—Heat Transfer by Conduction

1 From the **Multiphysics** menu, select **2 Heat Transfer by Conduction (ht)**.

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

3 Select Boundaries 1 and 4.

4 Set the **Boundary condition** to **Temperature**.

5 Enter `T0` in the **Temperature** edit field.

6 Select Boundaries 2, 3, 5, 7, 8, and 9.

7 Set the **Boundary condition** to **Heat flux**.

8 Enter `Te` in the **External temperature** edit field and `he` in the **Heat transfer coefficient** edit field.

   For the remaining external boundaries (number 10 and 11) keep the default setting, which is thermal insulation.

9 Click **OK**.

Boundary Conditions—Heat Transfer by Conduction 1

1 From the **Multiphysics** menu select **3 Heat Transfer by Conduction (ht1)**.

2 Select **Boundary Settings** from the **Physics** menu.

3 Select Boundaries 1 and 4.

4 Set the **Boundary condition** to **Temperature**.

5 Enter `0` in the **Temperature** edit field.

6 Select Boundaries 2, 3, 5, 7, 8, and 9.

7 Set the **Boundary condition** to **Heat flux**.

8 Enter `0` in the **External temperature** edit field and `he` in the **Heat transfer coefficient** edit field.

For the remaining external boundaries (number 10 and 11) keep the default setting, which is thermal insulation.

9 Click **OK**.

COMPUTING THE SOLUTION

First, compute the coupled linear response. Then perform the transient solution for the slow evolution of the beam temperature. Record the entire procedure in a solver script.

1 Choose **Solve>Solver Parameters** and change **Solver** to **Stationary**.

2 Set the **Linear system solver** to **Direct (PARDISO)**.

3 Click **OK**.

4 Go to **Solve>Solver Manager** and select **Initial value expression** from the **Initial value** area and **Use setting from Initial value frame** in the **Values of variables not solved for and linearization point** area.

5 On the **Solve For** page, select **Solid, Stress-Strain (smsld)** and **Heat Transfer by Conduction (ht1)**.

6 Click **Apply**.

7 On the **Script** page, click **Add Current Solver Settings**.

8 Click **OK**.

Continue with the transient solution set-up.

1 Go to **Solve>Solver Parameters** and change the **Solver** to **Time dependent**.

2 Select the **Allow complex numbers** check box because the source term involves complex valued variables from the frequency response analysis.

3 Enter `0:0.05:2` in the **Times** edit field.

4 Click **OK**.

5 Go to **Solve>Solver Manager**.

6 On the **Initial Value** page, select **Current solution** in both areas.

7 Go to the **Solve For** page and select **Heat Transfer by Conduction (ht)**.

8 Click **Apply**.

9 On the **Script** page, click **Add Current Solver Settings**.

10 Select the **Solve using a script** check box.

11 Click **OK**.

The solver script is ready. Before running it, set the excitation frequency.

1. From the **Physics** menu, choose **Scalar Variables**.
2. In the **Excitation frequency** edit field, type `7767`.
3. Click **OK** to close the **Application Scalar Variables** dialog box.
4. Click the **Solve** button on the Main toolbar to perform the computations.

POSTPROCESSING AND VISUALIZATION

To plot the temperature increase shown in Figure 15-27, follow these steps:

1. Click the **Plot Parameters** button on the Main toolbar.
2. In the **Plot type** area on the **General** page, select the **Subdomain**, **Deformed shape**, and **Geometry edges** check boxes, and clear all the others.
3. Click the **Subdomain** page. In the **Expression** edit field, enter `T-T0`.
4. Click **Apply** to plot the temperature increase at the end of the process, as shown in Figure 15-27 on page 706.

To visualize the temperature evolution in time, do as follows:

1. In the **Plot Parameters** dialog box, click the **Animate** tab.
2. Click the **Start Animation** button.

INDEX