Structural Mechanics Module

Verification Examples

5.3

COMSOL

Structural Mechanics Module Verification Examples

Version: COMSOL 5.3

Contact Information

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

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

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

Part number: CM021104

1

Introduction

This *Structural Mechanics Module Verification Manual* consists of a set of benchmark models from various areas of structural mechanics and solid mechanics engineering. These are models with a theoretical solution or an solution from an established benchmark. Their purpose is to show the close agreement between the numerical solution obtained in COMSOL Multiphysics and the established benchmark data, so that you can gain confidence in the solutions provided when using the Structural Mechanics Module.

The models illustrate the use of the various structural-mechanics specific physics interfaces and study types. We have tried to cover a wide spectrum of the capabilities in the Structural Mechanics Module.

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 models, we urge you to first read the *Structural Mechanics Module User's Guide*. This book introduces you to the functionality in the module, reviews new features, and covers basic modeling techniques with tutorials and example models. Another book, the *Structural Mechanics Module Applications Library*, contains a large number of examples models from important application areas such as automotive applications, dynamics and vibration, fluid-structure interaction, fatigue analysis, and piezoelectric applications.

For more information on how to work with the COMSOL Multiphysics graphical user interface, please refer to the *COMSOL Multiphysics Reference Manual* or the *Introduction to COMSOL Multiphysics* manual.

The book you are reading, the *Structural Mechanics Module Verification Manual*, provides details about a large number of ready-to-run models that provide numerical solutions to benchmark problems and textbook examples with theoretical closed-form solutions. Each entry comes with theoretical background, a discussion about the results with a comparison to the benchmark data or the analytical solution, as well as instructions that illustrate how to set it up. The documentation for all models contains references to the textbook or technical publication from which we have collected the benchmark data or other verification data.

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

Note: The full documentation set is available in electronic formats—PDF and HTML—through the COMSOL Documentation window after installation.

Comparison With Theoretical and Benchmark Results

COMSOL Multiphysics and the Structural Mechanics Module use the finite element method to solve problems on a computational mesh using discrete numerical methods. Theoretical, closed-form solutions are typically based on continuous mathematical models and would require infinitely small mesh elements to reproduce exactly. These benchmark models, on the other hand, use relatively coarse meshes. The comparisons of the numerical solution in COMSOL Multiphysics to the benchmark results therefore allow for a small discrepancy. Comparisons to established benchmark results also show similar accuracy. Sources to these differences in the results include different solution methods, different discretization (computational grids), and other differences between the code or method used in the benchmark and the COMSOL Multiphysics code. Also note that the numerical solution might vary slightly depending on the computer platform that you use because different platforms have small differences handling floating-point operations.

COMSOL Software Verification and Quality Assurance Programs

COMSOL uses extensive manual and automatic testing to validate and verify the code. The benchmark models in this book make up a subset of the test cases that are part of a continuous automatic testing program. The automatic test program also frequently rebuilds all models in the COMSOL Application Libraries to ensure that they work and provide consistent solutions.

Channel Beam

Introduction

In the following example you build and solve a simple 3D beam model using the 3D Beam interface. This example calculates the deformation, section forces, and stresses in a cantilever beam, and compares the results with analytical solutions. The first few natural frequencies are also computed. The purpose of the example is twofold: It is a verification of the functionality of the beam element in COMSOL Multiphysics, and it explains in detail how to give input data and interpret results for a nontrivial cross section.

This example also illustrates how to use the **Beam Cross Section** interface to compute the beam section properties and evaluate the stress distribution within the beam cross section.

Model Definition

The physical geometry is displayed in Figure 1. The finite element idealization consists of a single line.

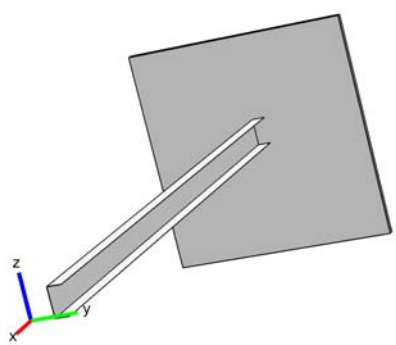


Figure 1: The physical geometry.

The cross section with its local coordinate system is shown in Figure 2. The height of the cross section is 50 mm and the width is 25 mm. The thickness of the flanges is 6 mm, while the web has a thickness of 5 mm. Note that the global y direction corresponds to the local negative z direction, and the global z direction corresponds to the local y direction. In the following, uppercase subscripts are used for the global directions and lowercase subscripts for the local directions.

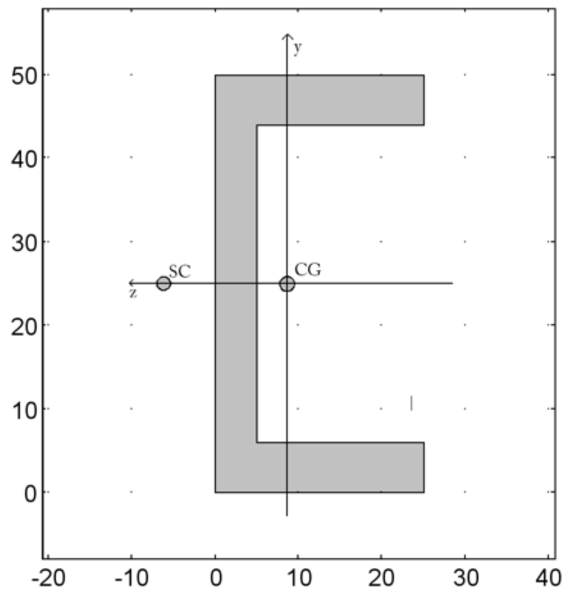


Figure 2: The beam cross section with local direction indicated.

For a detailed analysis, a case where the corners between the flange and the web are rounded are also studied. A 4 mm radius fillet is used at the external corner and a 2 mm radius fillet at the internal corner. This geometry is considered using the **Beam Cross Section** interface.

GEOMETRY

- Beam length, L = 1 m
- Cross-section area $A = 4.90 \cdot 10^{-4}$ m^2 (from the cross section library)
- Area moment of inertia in stiff direction, $I_{zz} = 1.69 \cdot 10^{-7}$ m^4
- Area moment of inertia in weak direction, $I_{yy} = 2.77 \cdot 10^{-8}$ m^4
- Torsional constant, $J = 5.18 \cdot 10^{-9}$ m^4
- Position of the shear center (SC) with respect to the area center of gravity (CG), $e_z = 0.0148$ m
- Torsional section modulus $W_\mathrm{t} = 8.64 \cdot 10^{-7}$ m^3
- Ratio between maximum and average shear stress for shear in y direction, μ_y=2.44

- Ratio between maximum and average shear stress for shear in z direction, μ_z=2.38
- Locations for axial stress evaluation are positioned at the outermost corners of the profile at the points (y_1, z_1)=(0.025, 0.0086), (y_2, z_2)=(0.025, −0.0164), (y_3, z_3)=(−0.025, −0.0164), and (y_4, z_4)=(−0.025, 0.0086) measured in the local coordinate system. The indices of the coordinates are point identifiers.

The values above are based on the idealized geometry with sharp corners. In a separate study you compute the section properties including fillets, using the **Beam Cross Section** interface.

MATERIAL

- Young's modulus, $E = 210$ GPa
- Poisson's ratio, $\nu = 0.25$
- Mass density, $\rho = 7800\ \mathrm{kg/m^3}$

CONSTRAINTS

One end of the beam is fixed.

LOADS

In the first load case, the beam is subjected to three forces and one twisting moment at the tip. The values are:

- Axial force $F_X = 10$ kN
- Transverse forces $F_Y = 50$ N and $F_Z = 100$ N
- Twisting moment $M_X = -10$ Nm

In the second load case, the beam is subjected to a gravity load in the negative Z direction.

The third case is an eigenfrequency analysis.

Results and Discussion

The analytical solutions for a slender cantilever beam with loads at the tip are summarized below. The displacements are

$$\delta_X = \delta_x = \frac{F_x L}{EA} = \frac{F_X L}{EA} =$$

$$\frac{10000\ \mathrm{N} \cdot 1\ \mathrm{m}}{2 \cdot 10^{11}\ \mathrm{Pa} \cdot 4.90 \cdot 10^{-4}\ \mathrm{m}^2} = 1.02 \cdot 10^{-4}\ \mathrm{m}$$

$$\delta_Z = -\delta_y = \frac{-F_y L^3}{3EI_{zz}} = \frac{F_Z L^3}{3EI_{zz}} =$$

$$\frac{100\ \mathrm{N} \cdot (1\ \mathrm{m})^3}{3 \cdot 2 \cdot 10^{11}\ \mathrm{Pa} \cdot 1.69 \cdot 10^{-7}\ \mathrm{m}^4} = 9.86 \cdot 10^{-4}\ \mathrm{m}$$

$$\delta_Y = -\delta_z = \frac{-F_z L^3}{3EI_{yy}} = \frac{F_Y L^3}{3EI_{yy}} =$$

$$\frac{50\ \mathrm{N} \cdot (1\ \mathrm{m})^3}{3 \cdot 2 \cdot 10^{11}\ \mathrm{Pa} \cdot 2.77 \cdot 10^{-8}\ \mathrm{m}^4} = 3.01 \cdot 10^{-3}\ \mathrm{m}$$

$$\theta_X = \theta_x = \frac{M_x L}{GJ} = \frac{M_X L}{GJ} =$$

$$\frac{-10\ \mathrm{Nm} \cdot 1\ \mathrm{m}}{\frac{2 \cdot 10^{11}\ \mathrm{Pa}}{2(1 + 0.25)} \cdot 5.18 \cdot 10^{-9}\ \mathrm{m}^4} = -2.41 \cdot 10^{-2}\ \mathrm{rad}$$

The stresses from the axial force, shear force, and torsion are constant along the beam, while the bending moment and bending stresses, are largest at the fixed end. The axial stresses at the fixed end caused by the different loads are computed as

$$\sigma_{x, F\mathrm{x}} = \frac{F_x}{A} = \frac{F_X}{A} = \frac{10000\ \mathrm{N}}{4.90 \cdot 10^{-4}\ \mathrm{m}^2} = 2.04 \cdot 10^7\ \mathrm{Pa}$$

$$\sigma_{x, M\mathrm{z}} = \frac{-M_z y}{I_{zz}} = \frac{-F_y L y}{I_{zz}} = \frac{F_Z L y}{I_{zz}} = \qquad (1)$$

$$\frac{100\ \mathrm{N} \cdot 1\ \mathrm{m}}{1.69 \cdot 10^{-7}\ \mathrm{m}^4} \cdot y = 5.92 \cdot 10^8\ \frac{\mathrm{Pa}}{\mathrm{m}} \cdot y$$

$$\sigma_{x, M\mathrm{y}} = \frac{M_y z}{I_{yy}} = \frac{-F_z L z}{I_{yy}} = \frac{-F_Y L z}{I_{yy}} = \qquad (2)$$

$$\frac{-50\ \mathrm{N} \cdot 1\ \mathrm{m}}{2.77 \cdot 10^{-8}\ \mathrm{m}^4} \cdot y = -1.81 \cdot 10^9\ \frac{\mathrm{Pa}}{\mathrm{m}} \cdot z$$

In Table 1 the stresses in the stress evaluation points are summarized after insertion of the local coordinates y and z in Equation 1 and Equation 2.

TABLE 1: AXIAL STRESSES IN MPA AT EVALUATION POINTS

Point	Stress from F_x	Stress from F_y	Stress from F_z	Total bending stress	Total axial stress
1	20.4	14.8	-29.7	-14.9	5.5
2	20.4	-14.8	-29.7	-44.5	-24.1
3	20.4	-14.8	15.6	0.8	21.2
4	20.4	14.8	15.6	30.4	50.8

Due to the shear forces and twisting moment there are also shear stresses in the section. In general, the shear stresses have a complex distribution, which depends strongly on the geometry of the actual cross section. The peak values of the shear stress contributions from shear forces are

$$\tau_{\mathrm{sy,\,max}} = \mu_{\mathrm{y}}\tau_{\mathrm{sy,\,mean}} = \mu_{\mathrm{y}}\frac{F_{\mathrm{y}}}{A} = \mu_{\mathrm{y}}\frac{F_{\mathrm{Z}}}{A} =$$

$$2.44\cdot\frac{100\ \mathrm{N}}{4.90\cdot 10^{-4}\ \mathrm{m}^2} = 2.44\cdot 2.04\cdot 10^5\ \mathrm{Pa} = 4.98\cdot 10^5\ \mathrm{Pa}$$

$$\tau_{\mathrm{sz,\,max}} = \mu_{\mathrm{z}}\tau_{\mathrm{sz,\,mean}} = \mu_{\mathrm{z}}\frac{F_{\mathrm{z}}}{A} = \mu_{\mathrm{z}}\frac{-F_{\mathrm{Y}}}{A} =$$

$$2.38\cdot\frac{-50\ \mathrm{N}}{4.90\cdot 10^{-4}\ \mathrm{m}^2} = -2.38\cdot 1.02\cdot 10^5\ \mathrm{Pa} = -2.43\cdot 10^5\ \mathrm{Pa}$$

The peak value of the shear stress created by torsion is

$$\tau_{\mathrm{t,\,max}} = \frac{|M_x|}{W_{\mathrm{t}}} = \frac{|M_X|}{W_{\mathrm{t}}} = \frac{10\ \mathrm{Nm}}{8.64\cdot 10^{-7}\ \mathrm{m}^3} = 11.6\cdot 10^6\ \mathrm{Pa}$$

Since the general cross-section data used for the analysis cannot predict the exact locations of the peak stresses from each type of action, a conservative scheme for combining the stresses is used in COMSOL Multiphysics. If the computed results exceeds allowable values somewhere in a beam structure, this may be due to this conservatism. You must then check the details, using information about the exact type of cross section and combination of loadings. This can be done using the **Beam Cross Section** interface.

The conservative maximum shear stresses are created by adding the maximum shear stress from torsion to the maximum shear stresses from shear force:

$$\tau_{xz,\,max} = |\tau_{sz,\,max}| + \tau_{t,\,max} = 11.8 \cdot 10^6 \text{ Pa}$$
$$\tau_{xy,\,max} = |\tau_{sy,\,max}| + \tau_{t,\,max} = 12.1 \cdot 10^6 \text{ Pa}$$

A conservative effective stress is then computed as

$$\sigma_{mises} = \sqrt{\sigma_{max}^2 + 3\tau_{xy,\,max}^2 + 3\tau_{xz,\,max}^2} = 58.6 \cdot 10^6 \text{ Pa}$$

The maximum normal stress, σ_{max}, is taken as the highest absolute value in the any of the stress evaluation points (the rightmost column in Table 1).

The COMSOL results for the first load case give 58.6 MPa von Mises stress at the constrained end of the beam which is in total agreement with the analytical solution. Actually, the results would have been the same with any mesh density, because the formulation of the beam elements in COMSOL contains the exact solutions to beam problems with only point loads.

In the second load case there is an evenly distributed gravity load. Since the resultant of a gravity load acts through the mass center of the beam, it does not just cause pure bending but also a twist of the beam. The reason is that in order to cause pure bending, a transverse force must act through the shear center of the section. In COMSOL Multiphysics this effect is automatically accounted for when you apply an edge load. An additional edge moment is created, using the e_z (or, depending on load direction, e_y) cross section property. The analytical solution to the tip deflections in the self-weight problem is

$$\delta_z = -\delta_y = \frac{-q_y L^4}{8EI_{zz}} = \frac{q_z L^4}{8EI_{zz}} = \frac{-\rho g A L^4}{8EI_{zz}} =$$
$$\frac{-8000\frac{\text{kg}}{\text{m}^3} \cdot 9.81\frac{\text{m}}{\text{s}^2} \cdot 4.90 \cdot 10^{-4}\text{ m}^2 \cdot (1\text{ m})^4}{8 \cdot 2 \cdot 10^{11}\text{ Pa} \cdot 1.69 \cdot 10^{-7}\text{ m}^4} = -1.42 \cdot 10^{-4}\text{ m}$$

$$\theta_x = \frac{m_x L^2}{2GJ} = \frac{q_y e_z L^2}{2GJ} = \frac{\rho g A e_z L^2}{2GJ} =$$

$$\frac{-8000\frac{\text{kg}}{\text{m}^3} \cdot 9.81\frac{\text{m}}{\text{s}^2} \cdot 4.90 \cdot 10^{-4}\text{ m}^2 \cdot 0.0148\text{ m} \cdot (1\text{ m})^2}{2 \cdot \frac{2 \cdot 10^{11}\text{ Pa}}{2(1+0.25)} \cdot 5.18 \cdot 10^{-9}\text{ m}^4} = -6.87 \cdot 10^{-2}\text{ rad}$$

Also for this case, the COMSOL Multiphysics solution captures the analytical solution exactly. Note, however, that in this case the resolution of the stresses is mesh dependent.

When using a shear center offset as in this example, you must bear in mind that the beam theory assumes that torsional moments and shear forces are applied at the shear center, while axial forces and bending moments are referred to the center of gravity. Thus, when point loads are applied it may be necessary to account for this offset.

The mode shapes and the natural frequencies of the beam are of three types: tension, torsion, and bending. The analytical expressions for the natural frequencies of the different types are:

$$f_{n,\,\mathrm{tension}} = \frac{2n+1}{4L}\sqrt{\frac{E}{\rho}} \tag{3}$$

$$f_{n,\,\mathrm{torsion}} = \frac{2n+1}{4L}\sqrt{\frac{GJ}{\rho(I_{\mathrm{yy}}+I_{\mathrm{zz}})}} \tag{4}$$

$$f_{n,\,\mathrm{bending}} = \frac{k_n}{2\pi}\sqrt{\frac{EI}{\rho A L^4}} \tag{5}$$

$$\cos(\sqrt{k_n})\cosh(\sqrt{k_n}) = -1$$

$$\Rightarrow\quad k_n = 3.516,\ 22.03,\ 61.70,\ 120.9,\ 200.0,\ \ldots$$

In Table 2 the computed results are compared with the results from Equation 3, Equation 4, and Equation 5. The agreement is generally very good. The largest difference occurs in Mode 12. This is the fifth order torsional mode, for which the mesh is not sufficient for a high accuracy resolution.

TABLE 2: COMPARISON BETWEEN ANALYTICAL AND COMPUTED NATURAL FREQUENCIES

Mode number	**Mode type**	**Analytical frequency (Hz)**	**COMSOL result (Hz)**
1	First y bending	21.02	21.04
2	First z bending	51.96	51.96
3	First torsion	128.3	128.4
4	Second y bending	131.7	131.8
5	Second z bending	325.5	325.7
6	Third y bending	368.8	369.2
7	Second torsion	384.9	388.4

TABLE 2: COMPARISON BETWEEN ANALYTICAL AND COMPUTED NATURAL FREQUENCIES

Mode number	Mode type	Analytical frequency (Hz)	COMSOL result (Hz)
8	Third torsion	641.5	658.1
9	Fourth y bending	722.8	724.1
10	Fourth torsion	898.1	943.7
11	Third z bending	911.8	912.0
12	Fifth torsion	1155	1251
13	Fifth y bending	1196	1199
14	First axial	1250	1251

When the computed section forces at the constrained end of the beam are fed into the **Beam Cross Section** interface, Figure 3 below shows the von Mises stress distribution within the cross section. One can notice that the maximum stress value is about 66 MPa which is slightly higher than the value computed in the beam interface (58 MPa). The stress computed with analytical cross section data is slightly underestimated. The reason is that the geometric representation used includes the fillets. If exactly the same cross section data are used, the stresses computed by the Beam interface are always conservative.

In Figure 4 to Figure 6 examples are shown of how the stress distributions from the individual section forces are displayed in the **Beam Cross Section** interface.

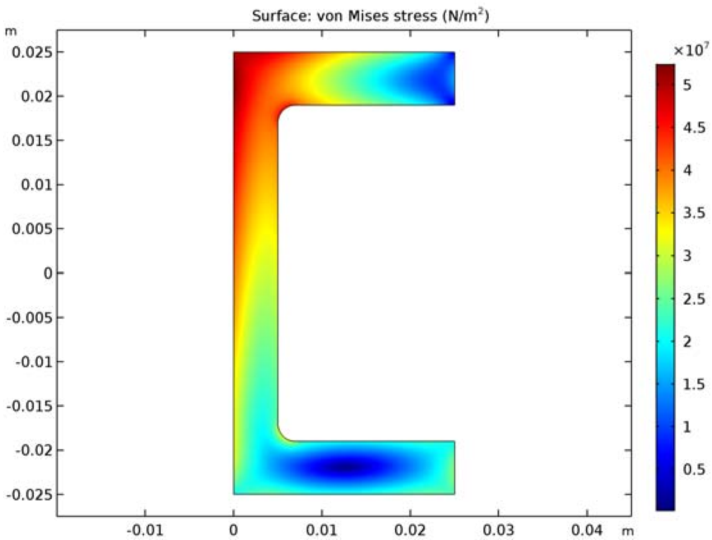


Figure 3: von Mises stress distribution at the fixed end (x = 0).

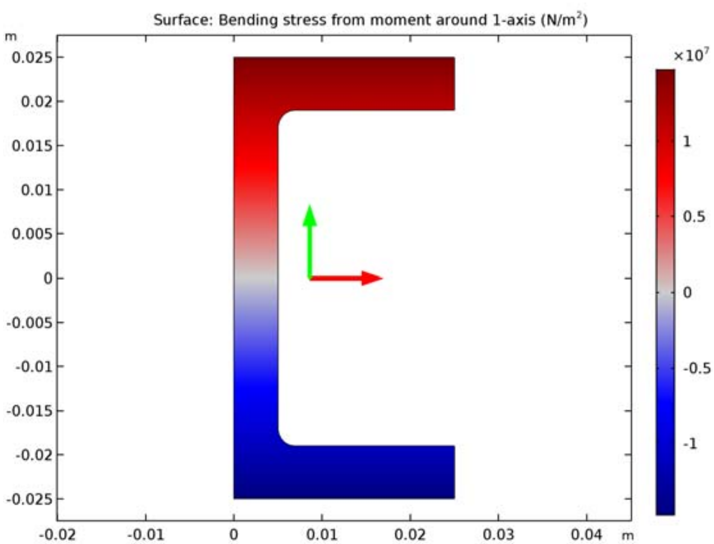


Figure 4: Plot of stresses from a bending moment. The center of gravity is highlighted.

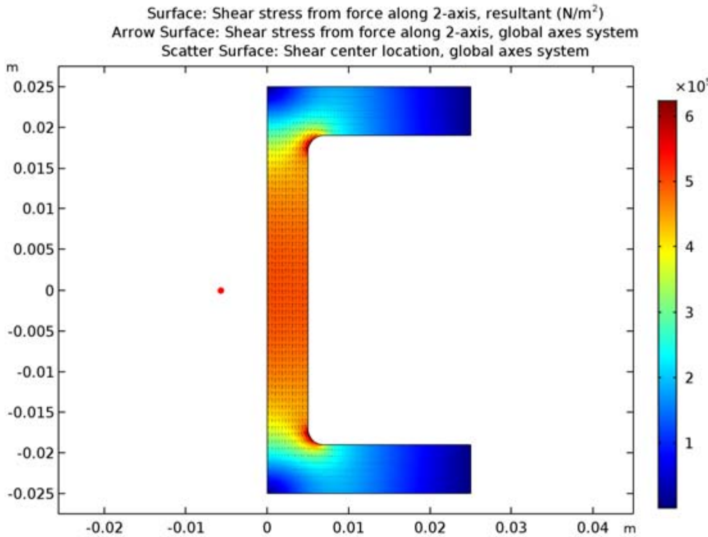


Figure 5: Plot of stresses from shear force. The shear center is highlighted.

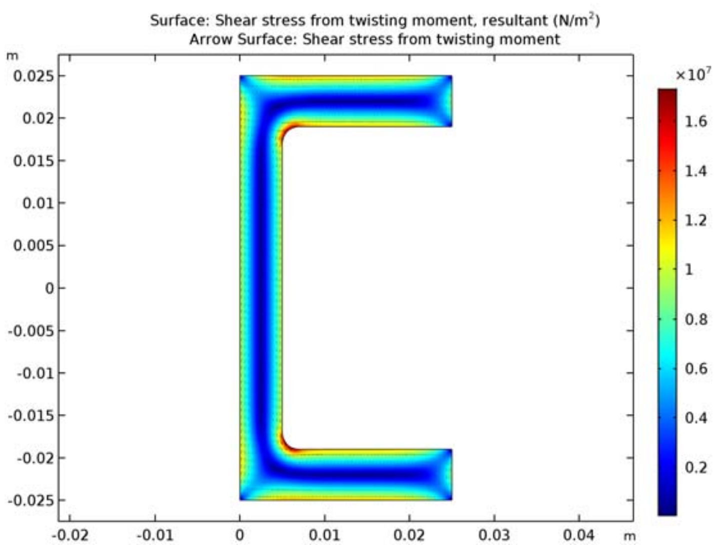


Figure 6: Plot of shear stresses from torsion.

Table 3 lists the beam cross section data computed using the **Beam Cross Section** interface and a geometry with fillets. There are significant differences in the maximum shear stress factor and torsional section modulus values. The stress concentration around the round corner explains these differences.

TABLE 3: COMPUTED BEAM CROSS SECTION DATA

Parameter	Value
Area	4.8485e-4 m^2
First moment of inertia	1.6556e-7 m^4
Distance to shear center in the first principal direction	0.014611 m
Second moment of inertia	2.7252e-8 m^4
Distance to shear center in the second principal direction	-9.5565e-9 m
Torsional constant	4.79754e-9 m^4
Torsional section modulus	5.6922e-7 m^3
Max shear stress factor in the second principal direction	3.0504
Max shear stress factor in the first principal direction	3.6711

If these cross section data are used in the Beam interface, the maximum von Mises stress is 73 MPa, which is slightly above the real value.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/channel_beam`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, click **3D**.
2. In the **Select Physics** tree, select **Structural Mechanics>Beam (beam)**.
3. Click **Add**.
4. Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`h1`	`25[mm]`	0.025 m	`Flange width`
`h2`	`50[mm]`	0.05 m	`Section height`
`t1`	`5[mm]`	0.005 m	`Web thickness`
`t2`	`6[mm]`	0.006 m	`Flange thickness`

4 In the **Model Builder** window, right-click **Global Definitions** and choose **Load Group**.

5 In the **Settings** window for **Load Group**, type `edge` in the **Parameter name** text field.

6 Right-click **Global Definitions** and choose **Load Group**.

7 In the **Settings** window for **Load Group**, type `point` in the **Parameter name** text field.

GEOMETRY 1

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **x** to `1`.

5 Click **Build All Objects**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`2e11`	Pa	Basic
Poisson's ratio	nu	`0.25`	1	Basic
Density	rho	8000	kg/m³	Basic

BEAM (BEAM)

Cross Section Data 1

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Beam (beam)** node, then click **Cross Section Data 1**.
2. In the **Settings** window for **Cross Section Data**, locate the **Cross Section Definition** section.
3. From the list, choose **Common sections**.
4. From the **Section type** list, choose **U-profile**.
5. In the h_y text field, type `h2`.
6. In the h_z text field, type `h1`.
7. In the t_y text field, type `t2`.
8. In the t_z text field, type `t1`.

Section Orientation 1

1. In the **Model Builder** window, expand the **Cross Section Data 1** node, then click **Section Orientation 1**.
2. In the **Settings** window for **Section Orientation**, locate the **Section Orientation** section.
3. From the **Orientation method** list, choose **Orientation vector**.
4. Specify the V vector as

0	x
0	y
1	z

Gravity 1

1. On the **Physics** toolbar, click **Edges** and choose **Gravity**.
2. Click in the **Graphics** window and then press Ctrl+A to select all edges.
3. On the **Physics** toolbar, click **Load Group** and choose **Load Group 1**.

Fixed Constraint 1

1. On the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.
2. Select Point 1 only.

Point Load 1

1. On the **Physics** toolbar, click **Points** and choose **Point Load**.
2. Select Point 2 only.
3. In the **Settings** window for **Point Load**, locate the **Force** section.
4. Specify the $\mathbf{F}_\mathrm{P}$ vector as

10e3	x
50	y
100	z

5. Specify the $\mathbf{M}_\mathrm{P}$ vector as

-10	x
0	y
0	z

6. On the **Physics** toolbar, click **Load Group** and choose **Load Group 2**.

STUDY 1

Step 1: Stationary

1. In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2. In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.
3. Locate the **Study Extensions** section. Select the **Define load cases** check box.
4. Click **Add** twice to add two rows to the load case table.
5. In the table, enter the following settings:

Load case	edge	Weight	point	Weight
Point load		1.0	√	1.0
Edge load	√	1.0		1.0

6. On the **Home** toolbar, click **Compute**.

RESULTS

Stress (beam)

The first default plot shows the Von mises stress distribution for the second load case. You can switch to the first load case to evaluate von Mises stress distribution caused by the point load.

1 In the **Model Builder** window, under **Results** click **Stress (beam)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Load case** list, choose **Point load**.
4 On the **Stress (beam)** toolbar, click **Plot**.

The following steps illustrate how to evaluate the displacement and stress values in specific tables.

Point Evaluation 1

1 On the **Results** toolbar, click **Point Evaluation**.
2 In the **Settings** window for **Point Evaluation**, type `Displacement/Rotation` in the **Label** text field.
3 Select Point 2 only.
4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Displacement>Displacement field>u - Displacement field, x component**.
5 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Displacement>Displacement field>v - Displacement field, y component**.
6 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Displacement>Displacement field>w - Displacement field, z component**.
7 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Displacement>Rotation field>thx - Rotation field, X component**.
8 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
u	m	delta_x
v	m	delta_y

Expression	Unit	Description
w	m	delta_z
thx	rad	theta_x

9 Click **Evaluate**.

Table 1

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Table 1**.

2 In the **Settings** window for **Table**, type `Displacement/Rotation` in the **Label** text field.

Point Evaluation 2

1 On the **Results** toolbar, click **Point Evaluation**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

4 From the **Parameter selection (Load case)** list, choose **First**.

5 In the **Label** text field, type `Axial Stress from Fx`.

6 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>Stress variables at first evaluation point> beam.s1 - Normal stress at first evaluation point**.

7 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>Stress variables at second evaluation point> beam.s2 - Normal stress at second evaluation point**.

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>Stress variables at third evaluation point> beam.s3 - Normal stress at third evaluation point**.

9 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>Stress variables at fourth evaluation point> beam.s4 - Normal stress at fourth evaluation point**.

10 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
beam.s1	MPa	first point
beam.s2	MPa	second point
beam.s3	MPa	third point
beam.s4	MPa	fourth point

11 Click **Evaluate**.

Table 2

1 In the **Model Builder** window, under **Results>Tables** click **Table 2**.

2 In the **Settings** window for **Table**, type `Normal Stress from Fx` in the **Label** text field.

Point Evaluation 3

1 On the **Results** toolbar, click **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, type `Total Bending Stress` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (Load case)** list, choose **First**.

4 Select Point 1 only.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>Stress variables at first evaluation point> beam.sb1 - Bending stress at first evaluation point**.

6 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>Stress variables at second evaluation point> beam.sb2 - Bending stress at second evaluation point**.

7 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>Stress variables at third evaluation point> beam.sb3 - Bending stress at third evaluation point**.

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>Stress variables at fourth evaluation point> beam.sb4 - Bending stress at fourth evaluation point**.

9 Locate the **Expressions** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`beam.sb1`	`MPa`	`first point`
`beam.sb2`	`MPa`	`second point`
`beam.sb3`	`MPa`	`third point`
`beam.sb4`	`MPa`	`fourth point`

10 Click **Evaluate**.

Table 3

1 In the **Model Builder** window, under **Results>Tables** click **Table 3**.

2 In the **Settings** window for **Table**, type `Total Bending Stress` in the **Label** text field.

Point Evaluation 4

1 On the **Results** toolbar, click **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, type `Shear Stress` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (Load case)** list, choose **First**.

4 Select Point 1 only.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>beam.tsymax - Max shear stress from shear force, y direction**.

6 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>beam.tszmax - Max shear stress from shear force, z direction**.

7 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>beam.ttmax - Max torsional shear stress**.

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>beam.txymax - Max shear stress, y direction**.

9 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>beam.txzmax - Max shear stress, z direction**.

10 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
beam.tsymax	MPa	Max shear stress from shear force, y direction
beam.tszmax	MPa	Max shear stress from shear force, z direction
beam.ttmax	MPa	Max torsional shear stress
beam.txymax	MPa	Max shear stress, y direction
beam.txzmax	MPa	Max shear stress, z direction

11 Click **Evaluate**.

Table 4

1 In the **Model Builder** window, under **Results>Tables** click **Table 4**.

2 In the **Settings** window for **Table**, type `Shear Stress` in the **Label** text field.

Perform an eigenfrequency analysis.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Eigenfrequency**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Eigenfrequency

Before computing the study, increase the desired number of eigenfrequencies.

1 In the **Model Builder** window, under **Study 2** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type `20`.

5 On the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (beam)

1 In the **Model Builder** window, under **Results** click **Mode Shape (beam)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **51.96**.

4 On the **Mode Shape (beam)** toolbar, click **Plot**.

The following steps illustrate how to use the Beam Cross Section interface to compute beam physical properties and evaluate stresses within a cross section.

Derived Values

Start by evaluating the section forces at the fixed end of the beam. These values are needed to get an accurate stress distribution within the beam cross section.

Point Evaluation 5

1 On the **Results** toolbar, click **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, type `Section Forces` in the **Label** text field.

3 Select Point 1 only.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Section forces>beam.Nxl - Local axial force**.

5 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Section forces>beam.Mzl - Bending moment, local z direction**.

6 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Section forces>beam.Tyl - Shear force, local y direction**.

7 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Section forces>beam.Myl - Bending moment, local y direction**.

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Section forces>beam.Tzl - Shear force, local z direction**.

9 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Section forces>beam.Mxl - Torsional moment, local x direction**.

10 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`beam.Nxl`	`N`	`N`
`beam.Mzl`	`N*m`	`M1`
`beam.Tyl`	`N`	`T2`
`beam.Myl`	`N*m`	`M2`
`beam.Tzl`	`N`	`T1`
`beam.Mxl`	`N*m`	`Mt`

11 Click **Evaluate**.

Table 5

1 In the **Model Builder** window, under **Results>Tables** click **Table 5**.

2 In the **Settings** window for **Table**, type `Section Forces` in the **Label** text field.

ROOT

On the **Home** toolbar, click **Component** and choose **Add Component>2D**.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Structural Mechanics>Beam Cross Section (bcs)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1** and **Study 2**.

5 Click **Add to Component** in the window toolbar.

6 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Beam (beam)** interface.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

Use the predefined Generic C-beam geometry part to draw the beam section geometry.

PART LIBRARIES

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

2 In the **Part Libraries** window, On the **Home** toolbar, click **Windows** and choose **Part Libraries**.

3 select **Structural Mechanics Module>Beams>Generic>C beam generic** in the tree.

4 Click **Add to Geometry**.

GEOMETRY 2

Generic C-beam 1 (pi1)

1 In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2** click **Generic C-beam 1 (pi1)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
d	h2	0.05 m	Beam height
b	h1	0.025 m	Flange width
tw	t1	0.005 m	Web thickness
tf	t2	0.006 m	Flange thickness

Name	Expression	Value	Description
r1	`2[mm]`	0.002 m	Web fillet radius
r2	0	0 mm	Flange fillet radius
slope	0	0	Flange slope [%]
u	0	0 mm	Flange thickness evaluation location

Form Union (fin)

1 In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2** right-click **Form Union (fin)** and choose **Build Selected**.

2 Click the **Zoom Extents** button on the **Graphics** toolbar.

3 Right-click **Form Union (fin)** and choose **Build Selected**.

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

BEAM CROSS SECTION (BCS)

On the **Physics** toolbar, click **Beam (beam)** and choose **Beam Cross Section (bcs)**.

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Beam Cross Section (bcs)**.

2 In the **Settings** window for **Beam Cross Section**, locate the **Section Forces** section.

3 In the N text field, type `1e4`.

4 In the M_1 text field, type `100`.

5 In the T_2 text field, type `100`.

6 In the M_2 text field, type `50`.

7 In the T_1 text field, type `-50`.

8 In the M_t text field, type `-10`.

STUDY 3

On the **Home** toolbar, click **Compute**.

Evaluate the beam physical properties required for the beam interface.

RESULTS

Section Properties

1 In the **Model Builder** window, under **Results>Derived Values** click **Section Properties**.

2 In the **Settings** window for **Global Evaluation**, click **New Table**.

BEAM (BEAM)

On the **Physics** toolbar, click **Beam Cross Section (bcs)** and choose **Beam (beam)**.

In the **Model Builder** window, under **Component 1 (comp1)** click **Beam (beam)**.

Cross Section Data 2

1. On the **Physics** toolbar, click **Edges** and choose **Cross Section Data**.
2. Click in the **Graphics** window and then press Ctrl+A to select all edges.
3. In the **Settings** window for **Cross Section Data**, locate the **Basic Section Properties** section.
4. In the A text field, type `comp2.bcs.A`.
5. In the I_{zz} text field, type `comp2.bcs.I1`.
6. In the e_z text field, type `comp2.bcs.ei1`.
7. In the I_{yy} text field, type `comp2.bcs.I2`.
8. In the e_y text field, type `comp2.bcs.ei2`.
9. In the J text field, type `comp2.bcs.J`.
10. Click to expand the **Stress evaluation properties** section. Locate the **Stress Evaluation Properties** section. In the h_y text field, type `comp2.bcs.h2`.
11. In the h_z text field, type `comp2.bcs.h1`.
12. In the w_t text field, type `comp2.bcs.Wt`.
13. In the μ_y text field, type `comp2.bcs.mu2`.
14. In the μ_z text field, type `comp2.bcs.mu1`.

Section Orientation 1

1. In the **Model Builder** window, expand the **Cross Section Data 2** node, then click **Section Orientation 1**.
2. In the **Settings** window for **Section Orientation**, locate the **Section Orientation** section.
3. Specify the P vector as

0	x
0	y
1	z

ADD STUDY

1. On the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.
4. Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Beam Cross Section (bcs)** interface.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

Some cross section properties are now defined using a dependent variable from the Beam Cross Section Interface. An example is the torsional section modulus defined as `comp2.bcs.Wt`. Follow the steps below to get access to these variables in this study.

STUDY 4

Step 1: Stationary

1 In the **Model Builder** window, under **Study 4** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Values of dependent variables** section.

3 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Study 3, Stationary**.

6 Locate the **Study Extensions** section. Select the **Define load cases** check box.

7 Click **Add**.

8 In the table, enter the following settings:

Load case	edge	Weight	point	Weight
`Point Load`		`1.0`	√	`1.0`

9 On the **Home** toolbar, click **Compute**.

RESULTS

Stress (beam) 1

1 In the **Model Builder** window, under **Results** click **Stress (beam) 1**.

2 On the **Stress (beam) 1** toolbar, click **Plot**.

STUDY 1

Compare the von Mises stress for the two cross sections.

Point Evaluation 6

On the **Results** toolbar, click **Point Evaluation**.

RESULTS

Point Evaluation 6

1 In the **Settings** window for **Point Evaluation**, type `Von Mises Stress` in the **Label** text field.

2 Locate the **Data** section. From the **Parameter selection (Load case)** list, choose **First**.

3 Select Point 1 only.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Beam>Stress>beam.mises - von Mises stress**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`beam.mises`	`MPa`	`von Mises stress`

6 Click **Evaluate**.

7 Locate the **Data** section. From the **Data set** list, choose **Study 4/Solution 4 (5) (sol4)**.

8 Click **Evaluate**.

Table 7

1 In the **Model Builder** window, under **Results>Tables** click **Table 7**.

2 In the **Settings** window for **Table**, type `Von Mises Stress` in the **Label** text field.

Finally modify **Study 1** and **Study 2** so that you can re-compute the solution later.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Beam (beam)> Cross Section Data 2**.

5 Click **Disable**.

STUDY 2

Step 1: Eigenfrequency

1 In the **Model Builder** window, expand the **Study 2** node, then click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Beam (beam)>Cross Section Data 2**.

5 Click **Disable**.

Created in COMSOL Multiphysics 5.3

Friction Between Contacting Rings

Introduction

This is a benchmark model involving stick-slip friction of a ring rolling inside another ring. The displacement of the inner ring is computed and compared to the analytical result (Ref. 1).

Model Definition

As illustrated in Figure 1, the geometry consists of two rings. The inner radius of the outer ring is 156 mm and a thickness of 4 mm. The inner ring has an inner radius of 100 mm and a thickness of 11.5 mm.

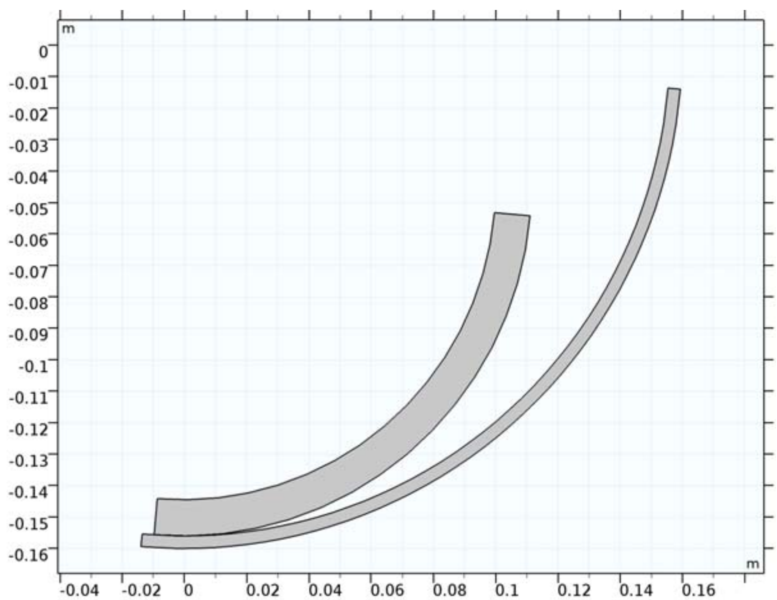


Figure 1: Model geometry.

The outer ring is rigid, which can be modeled by fully constraining its displacement. The inner ring is subjected to a prescribed rotation `phi` at its origin.

At the center of rotation, the resultant of the gravity load (P = 500 N) is applied to the inner ring.

A friction coefficient with the value 1 is used.

Results and Discussion

The analytical solution of the problem can be described as follows. The inner ring rolls along the outer ring until the tangential component of the gravity load becomes equal to the friction force (see Figure 2). At this critical point, slip occurs and the inner ring elevation reaches its maximum value.

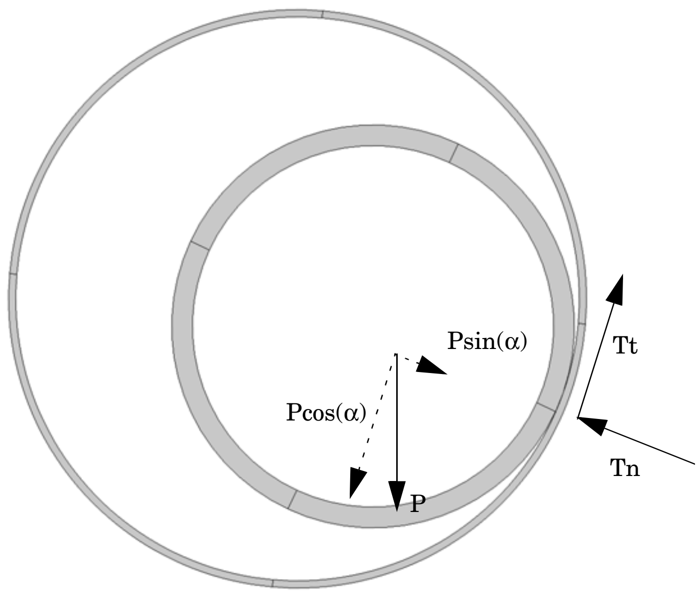


Figure 2: Representation of the contact and friction forces and the resultant of the gravity load.

The contact force corresponds to the normal component of the gravity load, $Tn = P\sin(\alpha)$. In this problem, the friction coefficient is 1, thus $Tn = Tt$ when sliding. As the critical position is reached when $Tt = P\cos(\alpha)$, the critical angle is $\alpha = 45^{\circ}$.

The maximum rolling distance is then $L = R \cdot \pi/4 = 122.5\text{ mm}$.

The vertical displacement of the center of the inner ring is defined as $Y = (R-r)(1-\cos(\alpha))$, where R is the inner radius of the outer ring and r is the outer radius of the inner ring. The maximum vertical displacement $Y_{max} = 13$ mm is reached at $\alpha = 45^{\circ}$.

[Figure 3](#) shows the von Mises stress distribution in the inner ring at the final step.

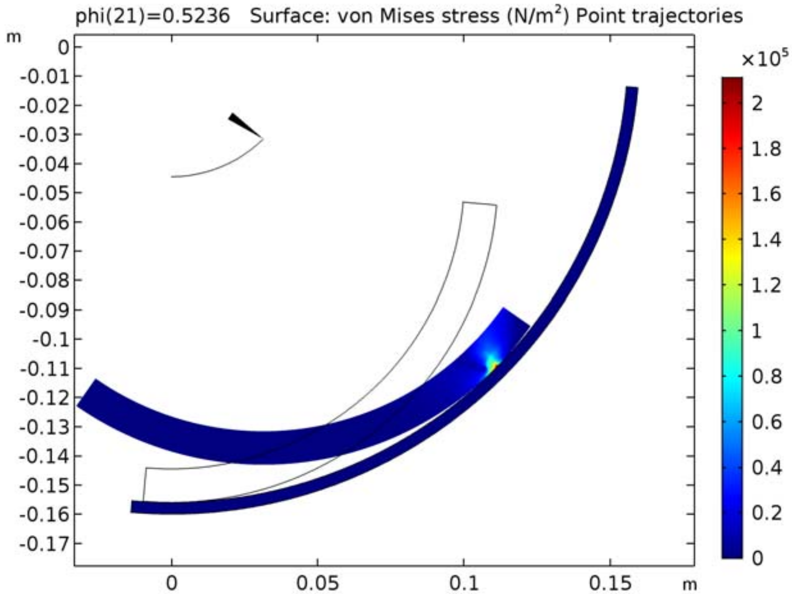


Figure 3: Stress distribution.

In Figure 4, you can see the elevation of the center of the inner ring with respect its rotation angle.

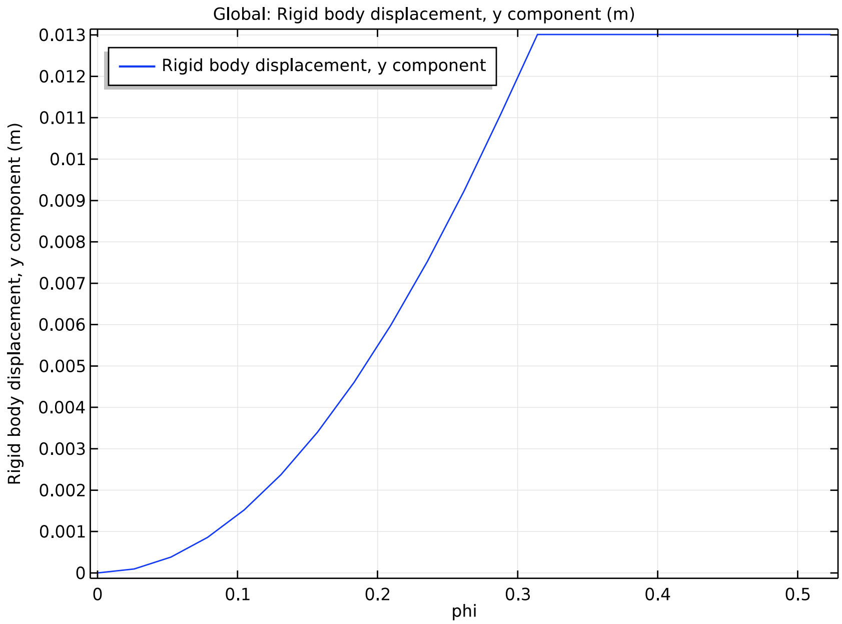


Figure 4: Elevation of the inner ring center versus applied rotation angle.

In agreement with the analytical solution, the computed maximum elevation is about 13 mm.

Figure 5 shows the contact pressure on the outer ring with respect to the ring curvature length. The peak of the contact pressure occurs at 123 mm as predicted by the analytical result.

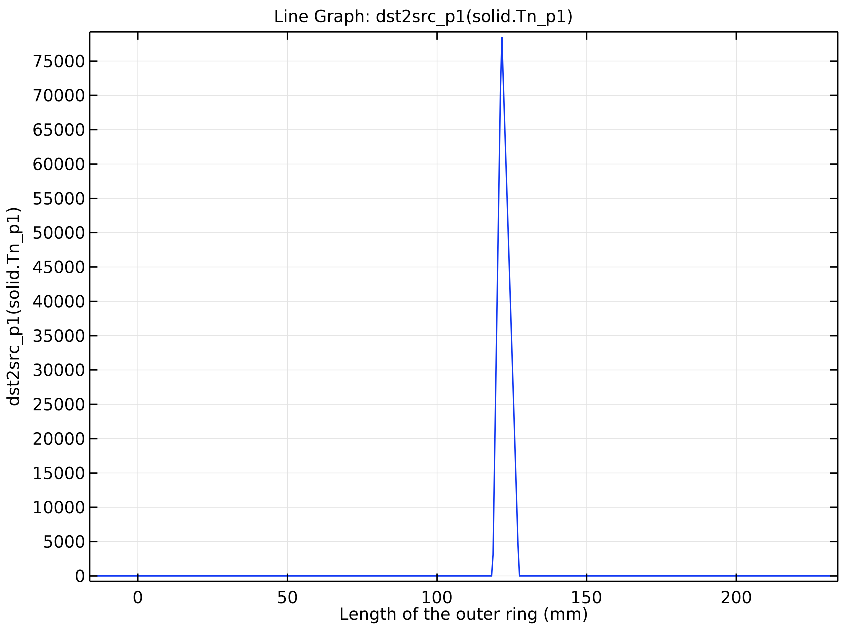


Figure 5: Contact pressure versus curvature length along the outer ring.

Notes About the COMSOL Implementation

A rigid connector is used to prescribed the rotation of the inner ring, while leaving the translation free so that it can follow the outer ring curvature. The rigid connector is attached to the inner boundary of the inner ring.

To capture the transition between stick friction and slip friction, a small continuation parameter step is used.

The model is not stable in its initial configuration; there are possible rigid body displacements before contact is established. To stabilize it, you add a small spring which is only active in the first parameter step.

Reference

1. Q. Feng and N.K. Prinja, "NAFEMS Benchmark Tests for Finite Element Modeling of Contact, Gapping and Sliding," *NAFEMS R0081*, 2001.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/contacting_rings`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
r1	160[mm]	0.16 m	Outer ring radius
r2	111.5[mm]	0.1115 m	Inner ring radius
y0	111.5[mm]-156[mm]	-0.0445 m	Inner ring center initial y-position

GEOMETRY 1

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r1`.

4 In the **Sector angle** text field, type `90`.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-95`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	4[mm]

7 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r2`.

4 In the **Sector angle** text field, type `90`.

5 Locate the **Position** section. In the **y** text field, type `y0`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-95`.

7 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	11.5[mm]

8 Right-click **Circle 2 (c2)** and choose **Build Selected**.

Delete Entities 1 (del1)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 On the object **c2**, select Domain 1 only.

5 On the object **c1**, select Domain 1 only.

6 Right-click **Component 1 (comp1)>Geometry 1>Delete Entities 1 (del1)** and choose **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Right-click **Component 1 (comp1)>Geometry 1>Form Union (fin)** and choose **Build Selected**.

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
L	156[mm]*(atan2(-y,-x)-pi/2)	m	Length of the outer ring

Contact Pair 1 (p1)

1 On the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 4 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

7 Select the **Active** toggle button.

8 Click **Paste Selection**.

9 In the **Paste Selection** dialog box, type 7 in the **Selection** text field.

10 Click **OK**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	210[GPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	7850	kg/m³	Basic

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 On the **Physics** toolbar, click **Domains** and choose **Fixed Constraint**.

2 Select Domain 1 only.

Contact 1

1 On the **Physics** toolbar, in the **Boundary** section, click **Pairs** and choose **Contact**.

2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.

3 In the **Pairs** list, select **Contact Pair 1 (p1)**.

Friction 1

1 On the **Physics** toolbar, click **Attributes** and choose **Friction**.

2 In the **Settings** window for **Friction**, locate the **Friction** section.

3 In the μ_{stat} text field, type 1.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
phi	0[rad]	0 rad	Inner ring rotation angle

SOLID MECHANICS (SOLID)

Rigid Connector 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

2 Select Boundary 8 only.

3 In the **Settings** window for **Rigid Connector**, locate the **Center of Rotation** section.

4 From the list, choose **User defined**.
5 Specify the $\mathbf{X}_c$ vector as

`0`	x
`y0`	y

6 Locate the **Prescribed Rotation** section. From the **By** list, choose **Prescribed rotation**.
7 In the ϕ_0 text field, type `-phi`.

Applied Force 1

1 Right-click **Rigid Connector 1** and choose **Applied Force**.
2 In the **Settings** window for **Applied Force**, locate the **Applied Force** section.
3 Specify the **F** vector as

`0`	x
`-500`	y

Spring Foundation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Rigid Connector 1** and choose **Spring Foundation**.
2 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.
3 In the $\mathbf{k}_\mathrm{u}$ text field, type `1e6*(phi==0)`.
4 Locate the **Rotational Spring** section. In the k_θ text field, type `1e6*(phi==0)`.

MESH 1

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.
2 Right-click **Mapped 1** and choose **Distribution**.
3 Select Boundary 5 only.
4 In the **Settings** window for **Distribution**, locate the **Distribution** section.
5 In the **Number of elements** text field, type `3`.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Boundary 7 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `60`.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `100`.

Distribution 4

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `1`.

5 Click **Build All**.

STUDY 1

Step 1: Stationary

Set up an auxiliary continuation sweep for the `phi` parameter.

1 In the **Settings** window for **Stationary**, click to expand the **Results while solving** section.

2 Locate the **Results While Solving** section. Select the **Plot** check box.

3 From the **Update at** list, choose **Steps taken by solver**.

4 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
phi	`range(0,pi/120,pi/6)`	

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Contact pressure (comp1.solid.Tn_p1)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `1e5`.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Dependent Variables 1** click **Friction force (spatial frame) (comp1.solid.Tt_p1)**.

7 In the **Settings** window for **Field**, locate the **Scaling** section.

8 In the **Scale** text field, type `1e5`.

9 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.

10 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

11 Select the **Tuning of step size** check box.

12 In the **Initial step size** text field, type `pi/1000`.

13 In the **Maximum step size** text field, type `pi/1000`.

14 In the **Minimum step size** text field, type `pi/10000`.

15 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1>Segregated 1** node, then click **Segregated Step 1**.

16 In the **Settings** window for **Segregated Step**, click to expand the **Method and termination** section.

17 Locate the **Method and Termination** section. In the **Number of iterations** text field, type `15`.

18 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Compile Equations: Stationary**.

19 In the **Settings** window for **Compile Equations**, click **Compute to Selected**.

RESULTS

Stress (solid)

Create a marker to make it easier to track the rotation of the inner ring. One way of doing it is to add an arrow to the default plot, which is generated below.

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

Point Trajectories 1

1 On the **Stress (solid)** toolbar, click **More Plots** and choose **Point Trajectories**.

2 In the **Settings** window for **Point Trajectories**, locate the **Trajectory Data** section.

3 In the **X-expression** text field, type `solid.u_rig1`.

4 In the **Y-expression** text field, type `y0+solid.v_rig1`.

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

6 In the **Arrow, X component** text field, type `cos(phi+5[deg])`.

7 In the **Arrow, Y component** text field, type `sin(-phi-5[deg])`.

8 From the **Arrow type** list, choose **Cone**.

9 From the **Arrow base** list, choose **Head**.

10 From the **Color** list, choose **Black**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

1D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Legend** section.

3 From the **Position** list, choose **Upper left**.

4 In the **Label** text field, type `Rigid body y-displacement`.

Global 1

1 Right-click **Rigid body y-displacement** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics> Rigid connectors>Rigid Connector 1>Rigid body displacement (spatial frame)>solid.rig1.v - Rigid body displacement, y component**.

3 On the **Rigid body y-displacement** toolbar, click **Plot**.

1D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Parameter selection (phi)** list, choose **Last**.

Line Graph 1

1 Right-click **1D Plot Group 3** and choose **Line Graph**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `dst2src_p1(solid.Tn_p1)`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6 In the **Expression** text field, type `L`.
7 From the **Unit** list, choose **mm**.
8 On the **1D Plot Group 3** toolbar, click **Plot**.

1D Plot Group 3

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.
2 In the **Settings** window for **1D Plot Group**, type `Contact pressure along outer ring` in the **Label** text field.

Created in COMSOL Multiphysics 5.3

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 tutorial includes a comparison with the COMSOL Multiphysics solution. The application is based on a NAFEMS benchmark (see Ref. 1).

Model Definition

This is a plane strain problem and the 2D Solid Mechanics interface from the Structural Mechanics Module is thus suitable. The 2D geometry is further cut in half at the vertical symmetry axis.

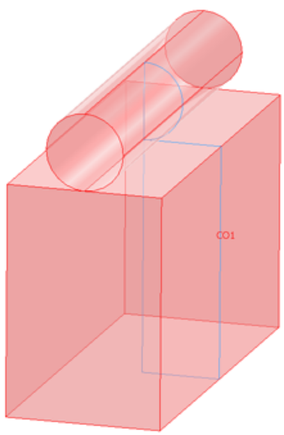


Figure 1: Model geometry.

In 2D, the cylinder is subjected to a point load along its top with an intensity of 35 kN/mm. Both the cylinder and block material are elastic, homogeneous, and isotropic.

The contact modeling method in this example only includes the frictionless part of the example described in Ref. 1. This model uses a contact pair, which is a straightforward way to implement a contact problem using the Solid Mechanics interface.

Figure 2 depicts the deformed shape and the von Mises stress distribution.

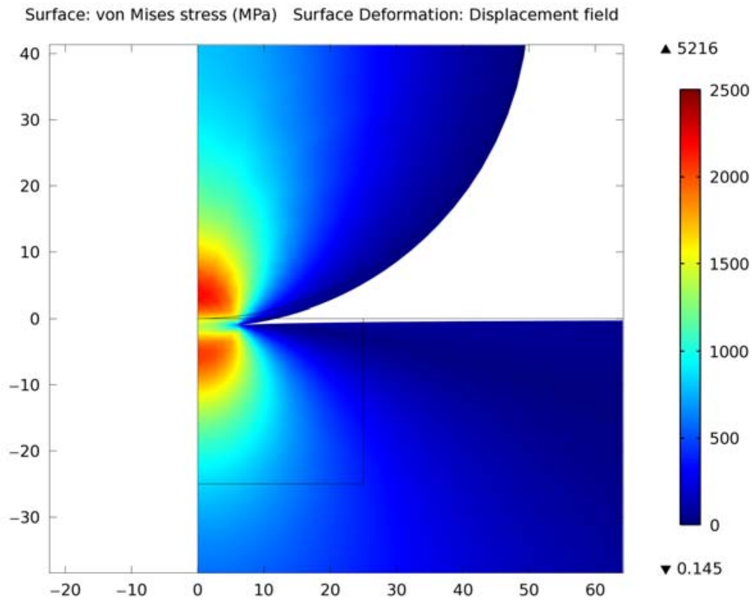


Figure 2: Deformation and von Mises stress at the contact area.

The analytical solution for the contact pressure as a function of the *x*-coordinate is

$$P = \sqrt{\frac{F_n E'}{2\pi R'} \times \left(1 - \left(\frac{x}{a}\right)^2\right)}$$

$$a = \sqrt{\frac{8F_n R'}{\pi E'}}$$

where F_n is the applied load per unit length, E' is the combined elasticity modulus, and R' is the combined radius. The combined Young's modulus and radius are defined as:

$$E' = \frac{2E_1E_2}{E_2(1-\nu_1^2) + E_1(1-\nu_2^2)}$$

$$R' = \lim_{R_2 \to \infty} \frac{R_1 R_2}{R_1 + R_2} = R_1$$

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. Combining these equations results in a contact length of 6.21 mm and a maximum contact pressure of 3585 MPa.

Figure 3 depicts the contact pressure along the contact area for both the analytical and the COMSOL Multiphysics solution.

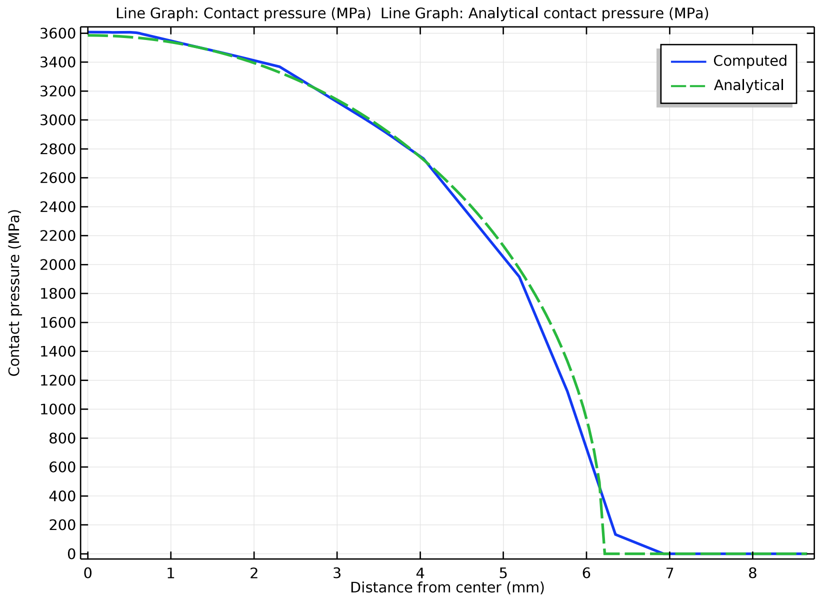


Figure 3: Analytical pressure distribution (solid line) and COMSOL Multiphysics solution (dashed line).

Notes About the COMSOL Implementation

The Structural Mechanics Module supports contact boundary conditions using contact pairs. The contact pair is defined by a source boundary and a destination boundary. The destination boundary is the one which is coupled to the source boundary if contact is established. The terms source and destination should be interpreted as in "the destination receives its displacements from the source." As a result, the contact pressure variable is available on the destination boundary. The mesh on the destination side should always be finer than on the source side.

In this example, the contact boundary pair consists of a flat source boundary and a curved destination boundary.

To reduce the number of iteration steps and improve convergence, it is good practice to set an initial contact pressure as close to the anticipated 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 example, it is necessary to specify an initial contact pressure to make the model stable with respect to the initial conditions, because the initial configuration—where the cylinder is free to move in the vertical direction—is singular. An alternative could be to define the geometries with a small overlap or supporting the roller with weak springs.

The small size of the contact region necessitates a local mesh refinement. Use a free mesh for the cylindrical domain and a mapped mesh for the aluminum block. The block geometry requires some modification to set up a refined mesh area.

The solver sequence set up as default by the program for a contact problem is a segregated solution, with displacements and contact pressures solved separately. The solver settings for the contact pressure step give optimal quality and should usually not be modified.

References

1. A.W.A. Konter, *Advanced Finite Element Contact Benchmarks*, NAFEMS, 2006.

2. M.A. Crisfield, *Non-linear Finite Element Analysis of Solids and Structures, volume 2: Advanced Topics*, John Wiley & Sons, London, 1997.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/cylinder_roller_contact`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
E1	70[GPa]	7E10 Pa	Block Young's modulus
E2	210[GPa]	2.1E11 Pa	Cylinder Young's modulus
nu0	0.3	0.3	Poisson's ratio
Fn	35[kN]	3.5E4 N	External load
E_star	2*E1*E2/((E1+E2)*(1-nu0^2))	1.154E11 Pa	Combined Young's modulus
R	50[mm]	0.05 m	Combined radius
d	200[mm]	0.2 m	Block width
th	1[mm]	0.001 m	Thickness
lc	10[mm]	0.01 m	Estimated contact length
a	sqrt(8*Fn*R/(pi*E_star*th))	0.006215 m	Analytical contact length
pmax	sqrt(Fn*E_star/(2*pi*R*th))	3.585E9 N/m²	Maximum contact pressure
dist	1[mm]	0.001 m	Initial distance between parts

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
p_analytical	pmax*sqrt(1-(x/a)^2)	N/m²	Analytical contact pressure

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Now create the geometry. Recall that you only need to model one half of the 2D cross section.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type R.

4 In the **Sector angle** text field, type 180.

5 Locate the **Position** section. In the **y** text field, type R+dist.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type -90.

7 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type d/2.

4 In the **Height** text field, type d.

5 Locate the **Position** section. In the **y** text field, type -d.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	d/2

7 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

Square 1 (sq1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `R/2`.

4 Locate the **Position** section. In the **y** text field, type `-R/2`.

5 Right-click **Square 1 (sq1)** and choose **Build Selected**.

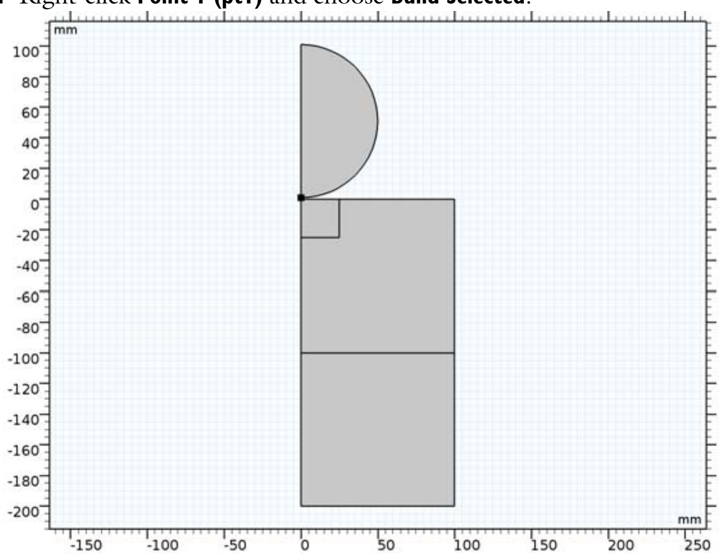
Point 1 (pt1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **y** text field, type `dist`.

4 Right-click **Point 1 (pt1)** and choose **Build Selected**.



Rotate 1 (rot1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **pt1** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation Angle** section.

4 In the **Rotation** text field, type `10`.

5 Locate the **Center of Rotation** section. In the **y** text field, type `R+dist`.

6 Right-click **Rotate 1 (rot1)** and choose **Build Selected**.

Convert to Solid 1 (csol1)

1 On the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 Right-click **Convert to Solid 1 (csol1)** and choose **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

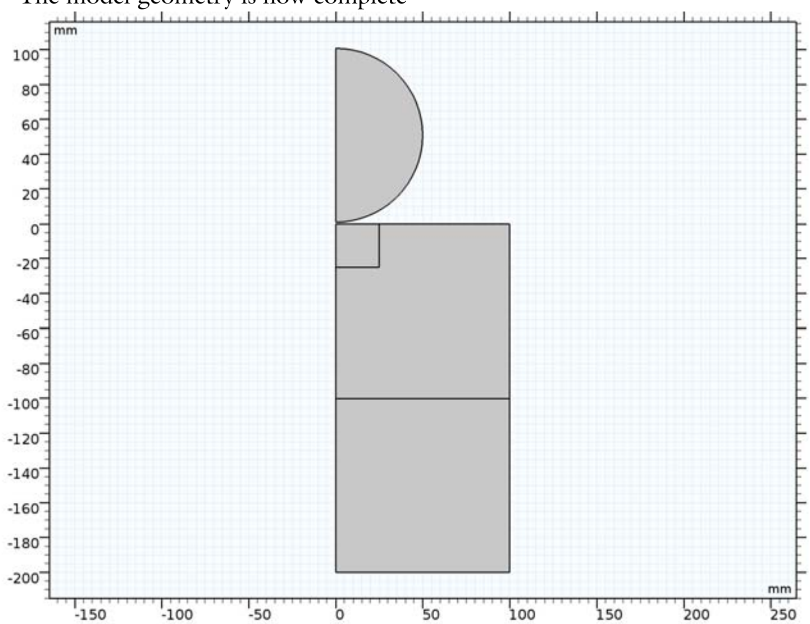
2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

5 Right-click **Component 1 (comp1)>Geometry 1>Form Union (fin)** and choose **Build Selected**.

The model geometry is now complete



DEFINITIONS

Contact Pair 1 (p1)

1 On the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundary 14 only.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Thickness** section.

3 In the d text field, type `th`.

Symmetry 1

1 Right-click **Component 1 (comp1)>Solid Mechanics (solid)** and choose **More Constraints> Symmetry**.

2 Select Boundaries 1, 3, 5, 8, and 9 only.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 Select Boundary 2 only.

Point Load 1

1 Right-click **Solid Mechanics (solid)** and choose **Points>Point Load**.

2 Select Point 7 only.

Use only half the total load since you model just one symmetry half of the full geometry.

3 In the **Settings** window for **Point Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_\mathrm{P}$ vector as

0	x
-Fn/2	y

Contact 1

1 Right-click **Solid Mechanics (solid)** and choose **Pairs>Contact**.

2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.

3 In the **Pairs** list, select **Contact Pair 1 (p1)**.

4 Locate the **Initial Values** section. In the T_n text field, type `(Fn/2)/(lc*th)`.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	E1	Pa	Basic
Poisson's ratio	nu	nu0	1	Basic
Density	rho	1	kg/m³	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 Select Domain 4 only.

3 In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	E2	Pa	Basic
Poisson's ratio	nu	nu0	1	Basic
Density	rho	1	kg/m³	Basic

The analytical solution to this problem assumes that engineering strains are used. Since the solution of a contact problem forces the study step to be geometrically nonlinear, you must explicitly enforce a linear strain representation.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Geometric Nonlinearity** section.

3 Select the **Force linear strains** check box.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.
2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domain 4 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 14 only.
5 Locate the **Element Size** section. Click the **Custom** button.
6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
7 In the associated text field, type `0.6`.
8 Click **Build All**.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.
2 Right-click **Mapped 1** and choose **Distribution**.
3 Select Boundaries 3, 6, and 10 only.
4 In the **Settings** window for **Distribution**, locate the **Distribution** section.
5 In the **Number of elements** text field, type `20`.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Boundary 1 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type `10`.
5 Click **Build All**.

Adjust the scale for the contact pressure variable based on the analytical solution.

STUDY 1

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 1** node, then click **Contact pressure (comp1.solid.Tn_p1)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `1e9`.

6 On the **Study** toolbar, click **Compute**.

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

4 On the **Stress (solid)** toolbar, click **Plot**.

Because the point load gives a singular stress at the top of the cylinder, adjust the color range to better see the stress distribution around the contact region.

5 Click to expand the **Range** section. Select the **Manual color range** check box.

6 In the **Maximum** text field, type `2500`.

7 On the **Stress (solid)** toolbar, click **Plot**.

Line Graph 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Model Builder** window, right-click **1D Plot Group 2** and choose **Line Graph**.

3 Select Boundary 14 only.

4 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics> Contact>solid.Tn - Contact pressure**.

5 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Component 1>Geometry>Coordinate (spatial frame)>x - x-coordinate**.

6 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`Computed`

10 On the **1D Plot Group 2** toolbar, click **Plot**.

Line Graph 2

1 Right-click **Results>1D Plot Group 2>Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables> p_analytical - Analytical contact pressure**.

3 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Analytical`

1D Plot Group 2

1 In the **Model Builder** window, under **Results** click **1D Plot Group 2**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** check box.

4 In the associated text field, type `Distance from center (mm)`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Contact pressure (MPa)`.

Line Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 2** click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 From the **Unit** list, choose **MPa**.

Line Graph 2

1 In the **Model Builder** window, under **Results>1D Plot Group 2** click **Line Graph 2**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 From the **Unit** list, choose **MPa**.

4 On the **1D Plot Group 2** toolbar, click **Plot**.

Stress Analysis of an Elliptic Membrane

General Description

In this benchmark, the static stress analysis described in the NAFEMS Test LE1, “Elliptic Membrane”, found on page 5 in Ref. 1 is performed. It is an analysis of a linear elastic plane stress model.

The computed stress level is compared with the values given in the benchmark report.

In addition to the original benchmark, a mesh convergence study is performed.

GEOMETRY

The geometry is an ellipse with an elliptical hole in it. The outer and inner edges are defined by the equations

$$\left(\frac{X}{3.25}\right)^2 + \left(\frac{Y}{2.75}\right)^2 = 1$$

$$\left(\frac{X}{2}\right)^2 + \left(\frac{Y}{1}\right)^2 = 1$$

The thickness (which actually does not influence the analysis) is 0.1 m.

Due to symmetry in load and in geometry, the analysis only includes a quarter of the geometry as shown in Figure 1.

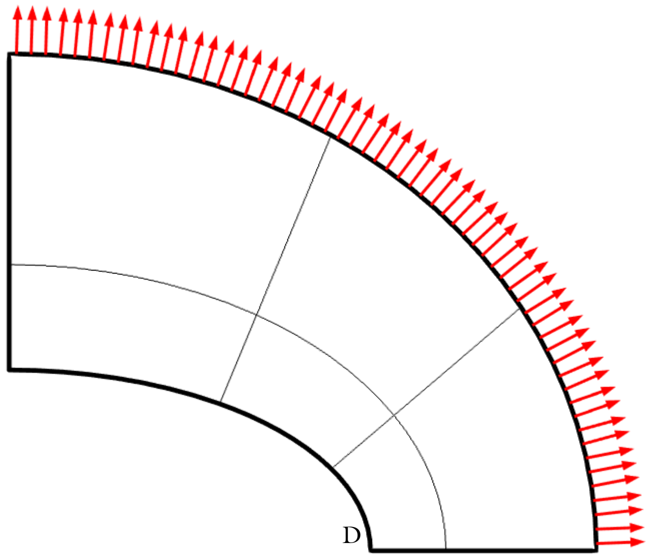


Figure 1: The geometry and load. Only the quarter which is analyzed is shown.

MATERIAL

Isotropic with $E = 2.1 \cdot 10^{11}$ Pa and $\nu = 0.3$.

LOAD

An evenly distributed load of 10 MPa acts along the outward normal of the outer boundary.

CONSTRAINTS

Symmetry conditions are used along the cuts at $X = 0$ and $Y = 0$.

Model Setup

The Solid Mechanics interface with the plane stress assumption is used.

Four meshes are exactly specified in Ref. 1. The 'coarse' mesh has 6 quadrilateral or 12 triangular elements. The 'fine' mesh has 24 quadrilateral or 48 triangular elements. The triangular elements are created by splitting the quadrilateral elements along a diagonal. The specified meshes are shown in Figure 2 and Figure 3.

For the mesh convergence study, these meshes are uniformly refined using a parameter `div`. The number of elements along the elliptical boundaries is `3*div` and the number of elements along the symmetry cuts is `2*div`.

The number of degrees of freedom varies from 48 (div = 1 and quadrilaterals with linear shape order) to 935810 (div = 64 and triangles with cubic shape order.)

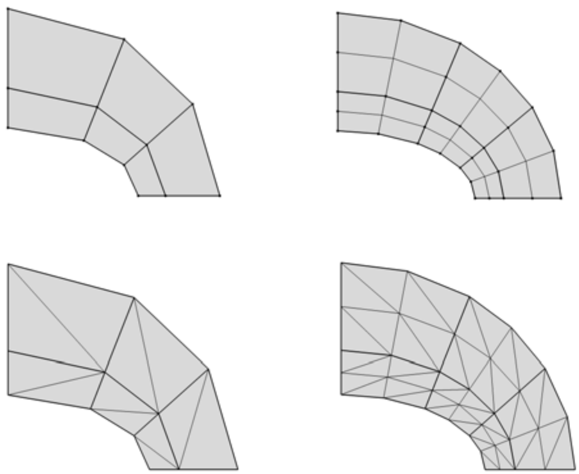


Figure 2: The meshes as specified in Ref. 1. Left column: 'coarse' (div=1). Right column: 'fine' (div=2).

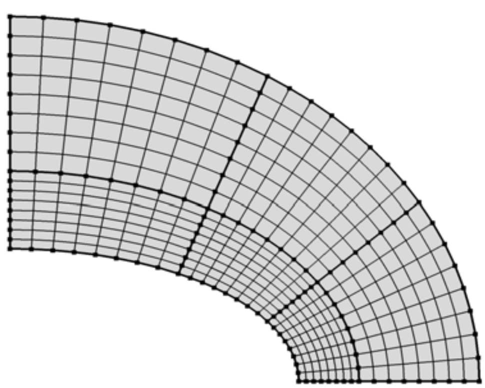


Figure 3: A quadrilateral mesh with div=8.

Due to the specification of the benchmark, the modeling differs somewhat from what you would use in practice:

- The internal boundaries in the model are created for matching the specification of the mesh in the NAFEMS benchmark as close as possible. If you were to solve the problem without these constraints, the modeling would be significantly simplified. Only two ellipses would be needed in the Geometry sequence.
- The knowledge about where a stress concentration is expected suggests that you should use a mesh such that more elements are present in the region around point D to get optimal accuracy, see Figure 1.
- Using the possibility to generate a free triangular mesh instead of one where quadrilateral elements are split along the diagonals would also give a mesh with better element quality.

Results and Discussion

The purpose of this test, in addition to a pure verification of the element formulation, is to check how well the software can represent a non-trivial geometrical shape such as an ellipse. It also evaluates the application of a distributed load.

The distribution of the direct stress in the Y direction is shown in Figure 4. As can be seen the result has steep gradients towards the point with maximum values.

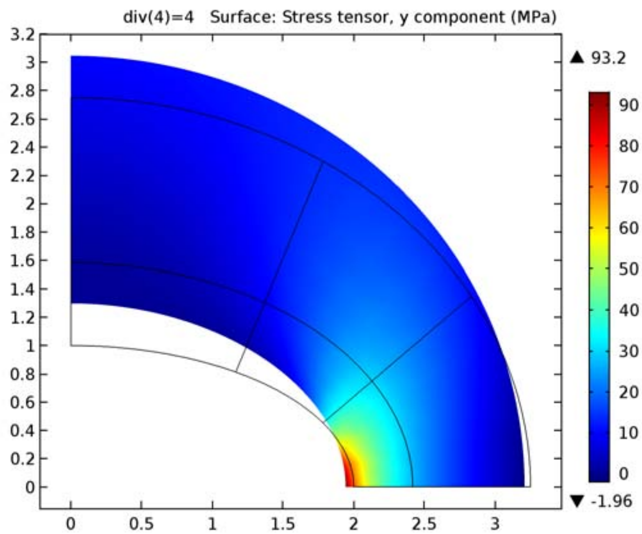


Figure 4: The distribution of the σ_y *stress component using div=4 and second order quadrilateral elements.*

The normal stress σ_y at the elliptic hole is evaluated at the point D located at $X = 2$, $Y = 0$ (see Figure 1). The target value according to Ref. 1 is 92.7 MPa. The value is based on an analytical result. The COMSOL results for the 'coarse' and 'fine' meshes are given in Table 1.

TABLE 1: COMPUTED RESULTS FOR THE MESHES SPECIFIED IN THE BENCHMARK

MESH	ELEMENT TYPE	DISCRETIZATION	COMPUTED VALUE	RELATIVE ERROR
Coarse	Triangle	Linear	55.7	-39.9%
Coarse	Quadrilateral	Linear	76.9	-17.0%
Fine	Triangle	Linear	75.2	-18.9%
Fine	Quadrilateral	Linear	88.2	-4.9%
Coarse	Triangle	Quadratic	76.6	-17.4%
Coarse	Quadrilateral	Quadratic	90.8	-2.0%
Fine	Triangle	Quadratic	88.9	-4.0%
Fine	Quadrilateral	Quadratic	93.3	0.6%
Coarse	Triangle	Cubic	95.9	3.5%
Coarse	Quadrilateral	Cubic	93.2	0.5%
Fine	Triangle	Cubic	93.4	0.8%
Fine	Quadrilateral	Cubic	92.8	0.1%

As can be expected, the coarse mesh is not able to capture the stress concentration unless elements with high order are used. Generally the quadrilaterals perform better than the corresponding triangles.

The mesh which is denoted as 'fine' is probably similar to what you would use in an analysis of a larger structure in a case where you are not specifically interested in a high resolution of the stress concentration. Still, with quadratic shape order elements the accuracy is good enough for most engineering purposes. Using elements with linear shape functions for structural analysis is commonly avoided in the finite element community.

The results of the mesh convergence study are shown in Figure 5. The element size h is defined as `0.417[m]/div`, which is the length of an edge in the element where the stress is measured.

The target value in Ref. 1, 92.7 MPa, is given with only three digits. This is not accurate enough for the convergence study here. Instead, the error is measured relative to the value 92.65817 MPa, towards which σ_y converges.

The convergence behavior is as expected since it is faster for elements with a higher shape function order. It can also be seen that quadrilaterals are somewhat more accurate than triangles for quadratic and cubic elements.

For the linear elements the triangles have a smaller error than the quadrilateral elements when element size decreases. This is a coincidence for the chosen evaluation point and is not valid generally. For a different point in the geometry the results are better for the quadrilateral elements.

The other two in-plane stress components σ_x and τ_{xy} should both be zero at point D since the boundary is free. In Figure 6 and Figure 7 similar convergence graphs are shown for these stress components.

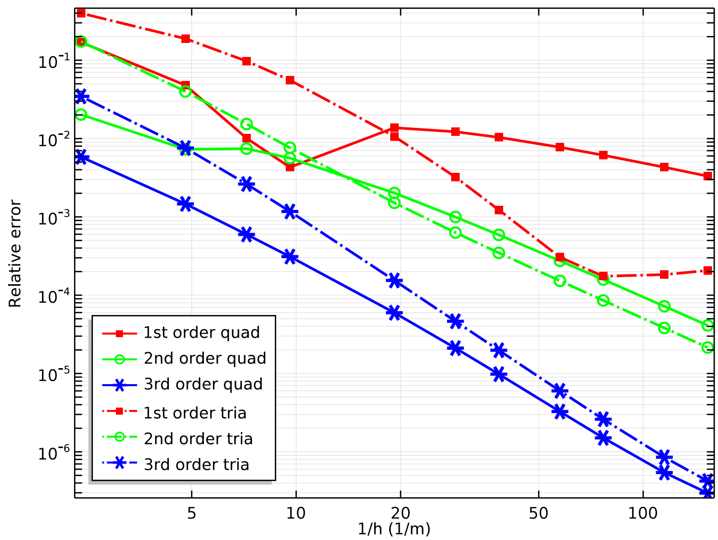


Figure 5: Error with respect to the stress target value as a function of the element size h.

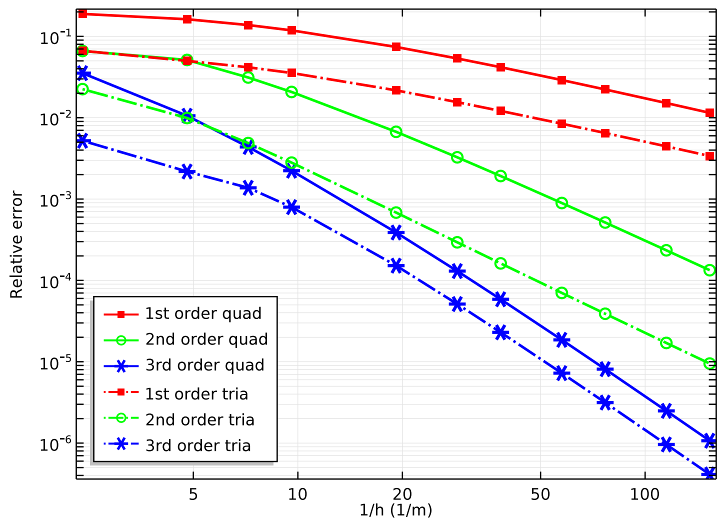


Figure 6: Error in the stress σ_x*. The values are normalized with the target for* σ_y*.*

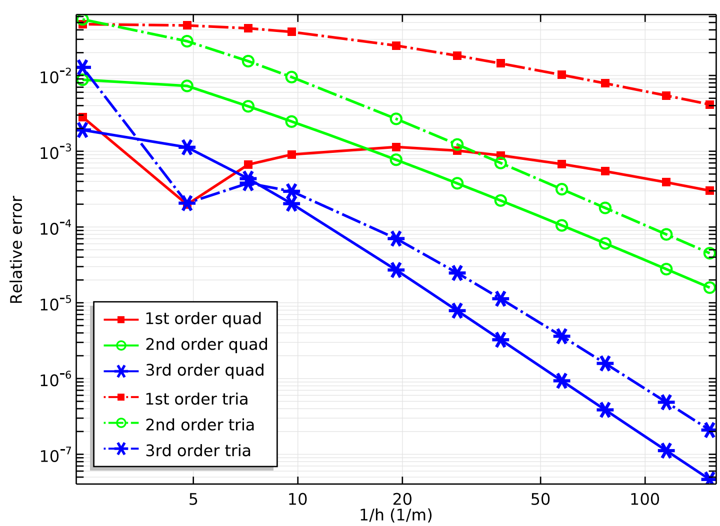


Figure 7: Error in the stress τ_{xy}*. The values are normalized with the target for* σ_y*.*

Since elements with different shape function orders are used, a comparison based only on element size may not be fair when efficiency is considered. The number of degrees of freedom in the model varies a lot for the same element size, and so does the solution time. In Figure 8, the error is shown as a function of the number of degrees of freedom. Also when compared this way, the elements with cubic shape functions have the best performance. This is usually true as long as the solutions are smooth, but it may not be true, for example, when solving nonlinear problems.

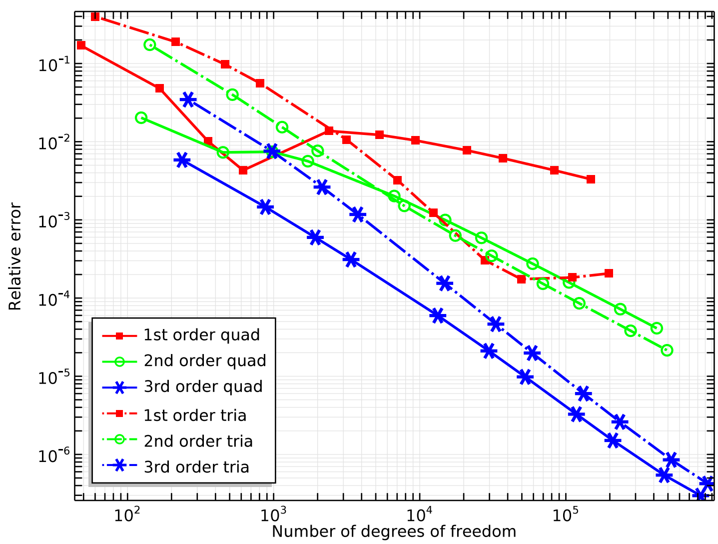


Figure 8: Error with respect to the stress target value as a function of the number of degrees of freedom.

Reference

1. G.A.O. Davies, R.T. Fenner, and R.W. Lewis, *Background to Benchmarks*, NAFEMS, Glasgow, 1993.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/elliptic_membrane`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.
2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
`div`	`1`	1	`Mesh refinement factor`
`sy_ref`	`92.65817[MPa]`	9.266E7 Pa	`Target stress`

GEOMETRY 1

Ellipse 1 (e1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Ellipse**.
2 In the **Settings** window for **Ellipse**, locate the **Size and Shape** section.
3 In the **Sector angle** text field, type `90`.
4 In the **a-semiaxis** text field, type `3.25`.
5 In the **b-semiaxis** text field, type `2.75`.

   Create an extra mesh control ellipse.

Ellipse 2 (e2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Ellipse**.

2 In the **Settings** window for **Ellipse**, locate the **Size and Shape** section.

3 In the **a-semiaxis** text field, type `2.417`.

4 In the **b-semiaxis** text field, type `1.583`.

5 In the **Sector angle** text field, type `90`.

Ellipse 3 (e3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Ellipse**.

2 In the **Settings** window for **Ellipse**, locate the **Size and Shape** section.

3 In the **a-semiaxis** text field, type `2`.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **e2** and **e1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **e3** only.

6 Click **Build All Objects**.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **General** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.

5 Find the **Control points** subsection. In row **1**, set **x** to `1.783` and **y** to `2.3`.

6 In row **2**, set **x** to `1.165` and **y** to `0.812`.

Bézier Polygon 2 (b2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **General** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.

5 Find the **Control points** subsection. In row **1**, set **x** to `2.833` and **y** to `1.348`.

6 In row **2**, set **x** to `1.783` and **y** to `0.453`.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`210E3[MPa]`	Pa	Basic
Poisson's ratio	nu	`0.3`	1	Basic
Density	rho	`0`	kg/m^3	Basic

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.

3 From the list, choose **Plane stress**.

4 Locate the **Thickness** section. In the d text field, type `0.1`.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1, 2, 9, and 11 only.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundaries 15, 18, and 21 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Pressure**.

5 In the p text field, type `-10[MPa]`.

MESH 1

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `div`.

5 Select Boundaries 1, 2, 13, 16, and 19 only.

6 Click **Build All**.

The default discretization of the displacement field is quadratic serendipity shape functions. Change to Lagrange shape functions.

7 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

SOLID MECHANICS (SOLID)

1 In the **Settings** window for **Solid Mechanics**, click to expand the **Discretization** section.

2 From the **Displacement field** list, choose **Quadratic Lagrange**.

Add also linear and cubic displacement fields. The actual selection of discretization type will be done in each study.

3 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Discretization 1

1 On the **Physics** toolbar, click **Global** and choose **Discretization**.

2 In the **Settings** window for **Discretization**, locate the **Discretization** section.

3 From the **Displacement field** list, choose **Linear**.

4 In the **Settings** window for **Discretization**, type `Discretization Linear` in the **Label** text field.

Discretization Linear 1

1 Right-click **Discretization Linear** and choose **Duplicate**.

2 In the **Settings** window for **Discretization**, locate the **Discretization** section.

3 From the **Displacement field** list, choose **Cubic Lagrange**.

4 In the **Settings** window for **Discretization**, type `Discretization Cubic` in the **Label** text field.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study Quad Linear` in the **Label** text field.

Parametric Sweep

On the **Study** toolbar, click **Parametric Sweep**.

STUDY QUAD LINEAR

Parametric Sweep

1 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

2 Click **Add**.

3 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
div	1 2 3 4 8 12 16 24 32 48 64	

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study Quad Linear** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Solid Mechanics	√	disc1

ROOT

Add five more studies for the other discretizations and element shapes. The parameter values are copied from the first study.

ADD STUDY

1 On the **Study** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

STUDY QUAD LINEAR

Parametric Sweep

In the **Model Builder** window, under **Study Quad Linear** right-click **Parametric Sweep** and choose **Copy**.

STUDY 2

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study Quad Quadratic` in the **Label** text field.

3 Right-click **Study Quad Quadratic** and choose **Paste Parametric Sweep**.

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

3 Click **Add Study** in the window toolbar.

STUDY 3

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study Quad Cubic` in the **Label** text field.

3 Right-click **Study Quad Cubic** and choose **Paste Parametric Sweep**.

STUDY QUAD CUBIC

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Solid Mechanics	√	disc2

MESH 1

Create a triangular mesh. This mesh case will be the default for the new studies created from now on.

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Duplicate**.

MESH 2

In the **Model Builder** window, expand the **Component 1 (comp1)>Meshes** node.

COMPONENT 1 (COMP1)

1 In the **Settings** window for **Mesh**, type `Mesh Tria` in the **Label** text field.

2 In the **Model Builder** window, expand the **Component 1 (comp1)>Meshes** node.

3 Right-click **Mesh Tria** and choose **More Operations>Convert**.

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

3 Click **Add Study** in the window toolbar.

STUDY 4

1 In the **Model Builder** window, click **Study 4**.

2 In the **Settings** window for **Study**, type `Study Tria Linear` in the **Label** text field.

3 Right-click **Study Tria Linear** and choose **Paste Parametric Sweep**.

STUDY TRIA LINEAR

Step 1: Stationary

1 In the **Settings** window for **Stationary**, click to expand the **Mesh selection** section.

2 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics interface	Solve for	Discretization
Solid Mechanics	√	disc1

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

3 Click **Add Study** in the window toolbar.

STUDY 5

1 In the **Model Builder** window, click **Study 5**.

2 In the **Settings** window for **Study**, type `Study Tria Quadratic` in the **Label** text field.

3 Right-click **Study Tria Quadratic** and choose **Paste Parametric Sweep**.

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

3 Click **Add Study** in the window toolbar.

4 On the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 6

1 In the **Model Builder** window, click **Study 6**.

2 In the **Settings** window for **Study**, type `Study Tria Cubic` in the **Label** text field.

3 Right-click **Study Tria Cubic** and choose **Paste Parametric Sweep**.

STUDY TRIA CUBIC

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Solid Mechanics	√	disc2

STUDY QUAD LINEAR

On the **Study** toolbar, click **Compute**.

STUDY QUAD QUADRATIC

1 In the **Model Builder** window, click **Study Quad Quadratic**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 On the **Study** toolbar, click **Compute**.

STUDY QUAD CUBIC

1 In the **Model Builder** window, click **Study Quad Cubic**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 On the **Study** toolbar, click **Compute**.

STUDY TRIA LINEAR

1 In the **Model Builder** window, click **Study Tria Linear**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 On the **Study** toolbar, click **Compute**.

STUDY TRIA QUADRATIC

1 In the **Model Builder** window, click **Study Tria Quadratic**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 On the **Study** toolbar, click **Compute**.

STUDY TRIA CUBIC

1 In the **Model Builder** window, click **Study Tria Cubic**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 On the **Study** toolbar, click **Compute**.

RESULTS

Point Graph 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Model Builder** window, right-click **1D Plot Group 2** and choose **Point Graph**.

3 Select Point 11 only.

4 In the **Settings** window for **Point Graph**, locate the **Data** section.

5 From the **Data set** list, choose **Study Quad Linear/Parametric Solutions 1 (sol2)**.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(solid.sy/sy_ref-1)`.

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type `div/0.417`.

9 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

10 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

11 From the **Positioning** list, choose **In data points**.

12 Click to expand the **Legends** section. Select the **Show legends** check box.

13 From the **Legends** list, choose **Manual**.

14 In the table, enter the following settings:

Legends
`1st order quad`

15 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

Point Graph 2

1 Right-click **Results>1D Plot Group 2>Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study Quad Quadratic/Parametric Solutions 2 (sol15)**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

6 Locate the **Legends** section. In the table, enter the following settings:

Legends
2nd order quad

Point Graph 3

1 Right-click **Results>1D Plot Group 2>Point Graph 2** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study Quad Cubic/Parametric Solutions 3 (sol28)**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

6 Locate the **Legends** section. In the table, enter the following settings:

Legends
3rd order quad

Point Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 2** click **Point Graph 1**.

2 In the Model Builder window, under **Results>1D Plot Group 2**, select **Point Graph 1**, **Point Graph 2**, and **Point Graph 3**. Then right-click and select **Duplicate**.

Point Graph 4

1 In the **Model Builder** window, under **Results>1D Plot Group 2** click **Point Graph 4**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study Tria Linear/Parametric Solutions 4 (sol41)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dash-dot**.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
1st order tria

Point Graph 5

1 In the **Model Builder** window, under **Results>1D Plot Group 2** click **Point Graph 5**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study Tria Quadratic/Parametric Solutions 5 (sol54)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dash-dot**.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
2nd order tria

Point Graph 6

1 In the **Model Builder** window, under **Results>1D Plot Group 2** click **Point Graph 6**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study Tria Cubic/Parametric Solutions 6 (sol67)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dash-dot**.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
3rd order tria

1D Plot Group 2

1 In the **Model Builder** window, under **Results** click **1D Plot Group 2**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** check box.

4 In the associated text field, type `1/h (1/m)`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Relative error`.

7 Click to expand the **Title** section. From the **Title type** list, choose **None**.

8 Click to expand the **Axis** section. Select the **x-axis log scale** check box.

9 Select the **y-axis log scale** check box.

10 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

11 In the **Model Builder** window, click **1D Plot Group 2**.

12 In the **Settings** window for **1D Plot Group**, type `Mesh convergence sy at D` in the **Label** text field.

13 On the **Mesh convergence sy at D** toolbar, click **Plot**.

Mesh convergence sy at D 1

1 Right-click **Mesh convergence sy at D** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Mesh convergence sx at D` in the **Label** text field.

Point Graph 1

1 In the **Model Builder** window, expand the **Results>Mesh convergence sx at D** node, then click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `abs(solid.sx/sy_ref)`.

4 Do the same modification for all graphs from **Point Graph 2** to **Point Graph 6**.

5 On the **Mesh convergence sx at D** toolbar, click **Plot**.

Mesh convergence sx at D 1

1 In the **Model Builder** window, under **Results** right-click **Mesh convergence sx at D** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Mesh convergence sxy at D` in the **Label** text field.

Point Graph 1

1 In the **Model Builder** window, expand the **Results>Mesh convergence sxy at D** node, then click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `abs(solid.sxy/sy_ref)`.

4 Do the same modification for all graphs from **Point Graph 2** to **Point Graph 6**.

5 On the **Mesh convergence sxy at D** toolbar, click **Plot**.

Mesh convergence sy at D 1

1 In the **Model Builder** window, under **Results** right-click **Mesh convergence sy at D** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Mesh convergence sy at D (by DOFs)` in the **Label** text field.

Point Graph 1

1 In the **Model Builder** window, expand the **Results>Mesh convergence sy at D (by DOFs)** node, then click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `12*div^2*1+10*div*1+2+6*div^2*4`.

Point Graph 2

1 In the **Model Builder** window, under **Results>Mesh convergence sy at D (by DOFs)** click **Point Graph 2**.

2 In the **Settings** window for **Point Graph**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `12*div^2*4+10*div*2+2+6*div^2*9`.

Point Graph 3

1 In the **Model Builder** window, under **Results>Mesh convergence sy at D (by DOFs)** click **Point Graph 3**.

2 In the **Settings** window for **Point Graph**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `12*div^2*9+10*div*3+2+6*div^2*16`.

Point Graph 4

1 In the **Model Builder** window, under **Results>Mesh convergence sy at D (by DOFs)** click **Point Graph 4**.

2 In the **Settings** window for **Point Graph**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `12*div^2*1+10*div*1+2+6*div^2*2*3`.

Point Graph 5

1 In the **Model Builder** window, under **Results>Mesh convergence sy at D (by DOFs)** click **Point Graph 5**.

2 In the **Settings** window for **Point Graph**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `12*div^2*4+10*div*2+2+6*div^2*2*6`.

Point Graph 6

1 In the **Model Builder** window, under **Results>Mesh convergence sy at D (by DOFs)** click **Point Graph 6**.

2 In the **Settings** window for **Point Graph**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `12*div^2*9+10*div*3+2+6*div^2*2*10`.

Mesh convergence sy at D (by DOFs)

1 In the **Model Builder** window, under **Results** click **Mesh convergence sy at D (by DOFs)**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 In the **x-axis label** text field, type `Number of degrees of freedom`.

4 On the **Mesh convergence sy at D (by DOFs)** toolbar, click **Plot**.

Point Evaluation 1

1 On the **Results** toolbar, click **Point Evaluation**.

2 Select Point 11 only.

3 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

4 From the **Data set** list, choose **Study Quad Linear/Parametric Solutions 1 (sol2)**.

5 From the **Parameter selection (div)** list, choose **From list**.

6 In the **Parameter values (div)** list, choose **1** and **2**.

7 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Solid Mechanics>Stress>Stress tensor (spatial frame)> solid.sy - Stress tensor, y component**.

8 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`solid.sy`	`MPa`	`Stress tensor, y component`

9 Click **Evaluate**.

Point Evaluation 2

1 Right-click **Point Evaluation 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

3 From the **Data set** list, choose **Study Quad Quadratic/Parametric Solutions 2 (sol15)**.

4 Click **Evaluate**.

Point Evaluation 3

1 Right-click **Results>Derived Values>Point Evaluation 2** and choose **Duplicate**.

2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

3 From the **Data set** list, choose **Study Quad Cubic/Parametric Solutions 3 (sol28)**.

4 Click **Evaluate**.

Point Evaluation 4

1 Right-click **Results>Derived Values>Point Evaluation 3** and choose **Duplicate**.

2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

3 From the **Data set** list, choose **Study Tria Linear/Parametric Solutions 4 (sol41)**.

4 Click **Evaluate**.

Point Evaluation 5

1 Right-click **Results>Derived Values>Point Evaluation 4** and choose **Duplicate**.

2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

3 From the **Data set** list, choose **Study Tria Quadratic/Parametric Solutions 5 (sol54)**.

4 Click **Evaluate**.

Point Evaluation 6

1 Right-click **Results>Derived Values>Point Evaluation 5** and choose **Duplicate**.

2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

3 From the **Data set** list, choose **Study Tria Cubic/Parametric Solutions 6 (sol67)**.

4 Click **Evaluate**.

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study Quad Quadratic/Parametric Solutions 2 (sol15)**.

4 From the **Parameter value (div)** list, choose **4**.

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Stress>Stress tensor (spatial frame)>solid.sy - Stress tensor, y component**.

3 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

4 On the **Stress (solid)** toolbar, click **Plot**.

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, click to expand the **Color legend** section.

3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

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

Failure Prediction in a Layered Shell

Introduction

Laminated shells made of carbon fiber reinforced plastic (CRFP) are common in a large variety of applications due to their high strength to weight ratio. Evaluation of the structural integrity of a laminated shell for a set of applied loads is necessary to make the design of such structures reliable.

This example shows how to model laminated shells using Shell interfaces in the Structural Mechanics Module**.** The structural integrity of a stack of shells with different fiber orientations is assessed through the parameters called Failure Index and Safety Factor, using different polynomial failure criteria. Because of the orientation, each ply will have different strength in the longitudinal and transversal direction, and hence different response to the loading. The analysis using a polynomial failure criterion is termed *first ply failure analysis*, where failure in any ply is considered as failure of the whole laminate. In this example, seven different polynomial criteria are compared.

The original model is a NAFEMS benchmark model, described in *Benchmarks for Membrane and Bending Analysis of Laminated Shells, Part 2: Strength Analysis* (Ref. 1). The COMSOL Multiphysics solutions are compared with the reference data.

Model Definition

The physical geometry of the problem consists of four square shells stacked above each other. The side length is 1 cm and each layer has thickness of 0.05 mm. The laminate (90/-45/45/0) is subjected to an in-plane axial tensile load. The actual geometry of the laminate is shown in Figure 1.

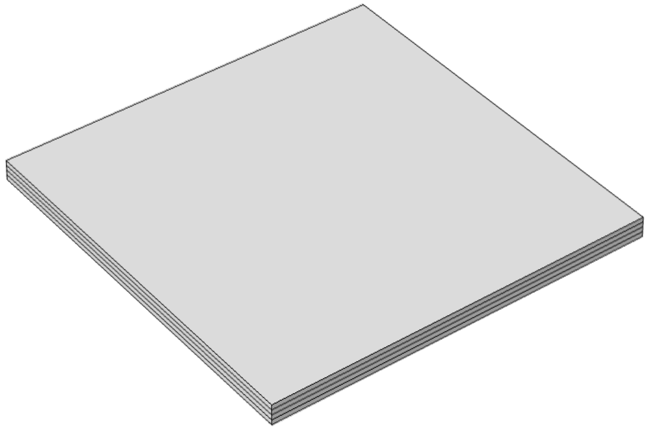


Figure 1: Geometry of layered shell with ply orientations 90/-45/45/0 from top to bottom.

MATERIAL PROPERTIES

The orthotropic material properties (Young's modulus, shear modulus, and Poisson's ratio) are given in Table 1:

TABLE 1: MATERIAL PROPERTIES

Material Property	**Value**
$\{E_1, E_2, E_3\}$	{207,7.6,7.6}(GPa)
$\{G_{12}, G_{23}, G_{13}\}$	{5,5,5}(GPa)
$\{\nu_{12}, \nu_{23}, \nu_{13}\}$	{0.3,0,0}

The tensile, compressive, and shear strengths are given in Table 2.

TABLE 2: MATERIAL STRENGTHS IN MPA

Material Strengths	**Value**
$\{\sigma_{t1}, \sigma_{t2}, \sigma_{t3}\}$	{500,5,5}(MPa)
$\{\sigma_{c1}, \sigma_{c2}, \sigma_{c3}\}$	{350,75,75}(MPa)
$\{\sigma_{ss23}, \sigma_{ss13}, \sigma_{ss12}\}$	{35,35,35}(MPa)

All material properties and strengths are given in the local material directions, where the first axis is aligned with the fiber orientation.

BOUNDARY CONDITIONS

The applied boundary conditions and loads on each node are given in the table below.

TABLE 3: NODE LOCATIONS AND BOUNDARY CONDITIONS.

Node	X [m]	Y [m]	Z [m]	Constrained DOF	Fx [N]	Fy [N]	Fz [N]
1(1)	0	0	0	x, y, z, θ_x, θ_y, θ_z	0	0	0
2(3)	0.01	0	0	θ_z	7.5	0	0
3(4)	0.01	0.01	0	θ_z	7.5	0	0
4(2)	0	0.01	0	x, θ_z	0	0	0

The numbers within parenthesis are point numbers in COMSOL Multiphysics geometry. The boundary conditions provided in the benchmark specifications apply to the layered shell as a single entity. The rotation around the z-axis, θ_z, is automatically constrained so it does not need to be considered.

FAILURE CRITERIA

Seven different failure criteria are used to predict the failure in the layered shell. These are Tsai-Wu Anisotropic, Tsai-Wu Orthotropic, Tsai-Hill, Hoffman, Modified Tsai-Hill, Azzi-Tsai-Hill, and Norris criteria.

Tsai-Wu Anisotropic

For the Tsai-Wu Anisotropic criterion, the material strength parameters are taken from Table 2 in order to obtain the same results as with the Tsai-Wu Orthotropic criterion. This exercise is done in order to verify the correctness of the implementation. The non-zero elements in the second rank tensor f are given below. Here, and in the following equations, repeated indices do not imply summation.

$$f_{ii} = \frac{1}{\sigma_{ti}} - \frac{1}{\sigma_{ci}}; \quad i = 1, 2, 3 \tag{1}$$

The non-zero elements in the fourth rank tensor F are

$$\begin{aligned} F_{ii} &= \frac{1}{\sigma_{ti}\sigma_{ci}}; \quad i = 1, 2, 3 \\ F_{44} &= \frac{1}{\sigma_{ss23}^2}, \quad F_{55} = \frac{1}{\sigma_{ss13}^2}, \quad F_{66} = \frac{1}{\sigma_{ss12}^2} \\ F_{ij} &= -\frac{1}{2}(\sqrt{F_{ii}F_{jj}}); \quad i = 1, 2, 3 \end{aligned} \tag{2}$$

Modified Tsai-Hill Orthotropic

The Hill criterion in Ref. 1 is called the Modified Tsai-Hill Orthotropic criterion in COMSOL Multiphysics.

Ref. 1 does not give results for neither the Tsai-Wu Anisotropic, Tsai-Hill, Azzi-Tsai-Hill, nor Norris criteria; so the analytical results for failure index and safety factor are here derived from the stress values given in Ref. 1.

The stresses from Ref. 1 are given in Table 4. Apart from σ_{11}, σ_{22}, and σ_{12}, all other stress components are either zero or negligible.

TABLE 4: STRESSES IN DIFFERENT PLIES.

Stresses	**Ply 1**	**Ply 2**	**Ply 3**	**Ply 4**
σ_{11}(MPa)	-5.128	12.59	8.520	9.357
σ_{22}(MPa)	4.407	1.983	0.125	-1.859
σ_{12}(MPa)	-1.663	2.572	-2.051	-0.5557

For all the selected polynomial criteria, the failure index (***FI***) is written as

$$FI = \sigma_i F_{ij} \sigma_j + \sigma_i f_i \tag{3}$$

where σ_i is the 6x1 stress vector (sorted using Voigt notation), $\boldsymbol{F}_{ij}$ is a 6x6 symmetric matrix (fourth rank tensor) that contains the coefficients for the quadratic terms, and f_i is a 6x1 vector (second rank tensor) that contains the linear terms. A failure index equal to or greater than 1.0 indicates failure in the material. In order to find the safety factor (***SF***), the applied stress in Equation 3 is multiplied by the safety factor SF, and the failure index ***FI*** is set equal to 1.0, which results in a quadratic equation of the form

$$a\,SF^2 + b\,SF = 1 \tag{4}$$

where $a = \sigma_i F_{ij} \sigma_j$ and $b = \sigma_i f_i$.

The lowest positive root in Equation 4 is selected as the safety factor (***SF***). Based on the stress values given in Table 4, the failure index and safety factor are computed for the criteria for which results in Ref. 1 are missing.

Tsai-Wu Anisotropic

For the Tsai-Wu Anisotropic criterion, the non-zero elements of the vector f_i and the matrix F_{ij} are given by Equation 1 and Equation 2. By taking values of stresses from Table 4, the failure index and safety factor are computed from Equation 3 and Equation 4, and given in Table 5 below.

TABLE 5: ANALYTIC VALUES OF FAILURE INDEX AND SAFETY FACTOR FOR TSAI-WU ANISOTROPIC CRITERION

Index	**Ply 1**	**Ply 2**	**Ply 3**	**Ply 4**
FI	0.8840	0.3730	0.0199	-0.34309
SF	1.122	2.536	14.30	31.88

Tsai-Hill Orthotropic

For the Tsai-Hill Orthotropic criterion, all elements of the vector f_i are zero, while the non-zero elements of matrix F_{ij} are given by Equation 5.

$$F_{ii} = \frac{1}{\sigma_{ti}^2}; \quad i = 1, 2, 3 \tag{5}$$

$$F_{44} = \frac{1}{\sigma_{ss23}^2}, \quad F_{55} = \frac{1}{\sigma_{ss13}^2}, \quad F_{66} = \frac{1}{\sigma_{ss12}^2}$$

$$F_{ij} = -\frac{1}{2}(F_{ii} + F_{jj} - F_{kk}); \quad i \neq j \neq k, i = 1, 2, 3$$

By taking values of stresses from Table 4, the failure index and safety factor are computed from Equation 3, Equation 4 and Equation 5, and given in Table 6 below.

TABLE 6: ANALYTIC VALUES OF FAILURE INDEX AND SAFETY FACTOR FOR TSAI-HILL CRITERION

Index	**Ply 1**	**Ply 2**	**Ply 3**	**Ply 4**
FI	0.7795	0.16323	0.0043	0.1390
SF	1.132	2.474	15.15	2.682

Azzi-Tsai-Hill

For the Azzi-Tsai-Hill criterion, all elements of the vector f_i are zero, while the non-zero elements of the matrix F_{ij} are given by Equation 6.

$$\begin{cases} \sigma_i \geq 0: & \left(F_{ii} = \dfrac{1}{\sigma_{ti}^2}\right) \\ \sigma_i < 0: & \left(F_{ii} = \dfrac{1}{\sigma_{ci}^2}\right) \end{cases} ; \quad i = 1, 2$$

$$F_{66} = \frac{1}{\sigma_{ss12}^2} \tag{6}$$

$$\begin{cases} \sigma_1 \geq 0: & \left(F_{12} = -\dfrac{1}{2\sigma_{t1}^2}\right) \\ \sigma_1 < 0: & \left(F_{12} = -\dfrac{1}{2\sigma_{c1}^2}\right) \end{cases}$$

By taking values of the stresses from Table 4, the failure index and safety factor are computed from Equation 3, Equation 4 and Equation 6, and given in Table 7 below .

TABLE 7: ANALYTIC VALUES OF FAILURE INDEX AND SAFETY FACTOR FOR AZZI-TSAI-HILL CRITERION

Index	**Ply 1**	**Ply 2**	**Ply 3**	**Ply 4**
FI	0.7796	0.1632	0.00435	0.00128
SF	1.132	2.474	15.15	27.87

Norris

For the Norris criterion, all elements of the vector f_i are zero, while the non-zero elements of the matrix F_{ij} are given by Equation 7.

$$\begin{cases} \sigma_i \geq 0: & \left(F_{ii} = \dfrac{1}{\sigma_{ti}^2}\right) \\ \sigma_i < 0: & \left(F_{ii} = \dfrac{1}{\sigma_{ci}^2}\right) \end{cases} ; \quad i = 1, 2$$

$$F_{66} = \frac{1}{\sigma_{ss12}^2} \tag{7}$$

$$F_{12} = -\frac{1}{2}\left(\sqrt{F_{11}F_{22}}\right)$$

By taking values of the stresses from Table 4, the failure index and safety factor are computed from Equation 3, Equation 4 and Equation 7, and given in Table 8 below .

TABLE 8: ANALYTIC VALUES OF FAILURE INDEX AND SAFETY FACTOR FOR NORRIS CRITERION

Index	Ply 1	Ply 2	Ply 3	Ply 4
FI	0.7923	0.1533	0.0039	0.00168
SF	1.126	2.553	15.95	24.38

Note that for the current model, failure index and safety factor are computed at the mid-plane of each shell interface. However, COMSOL Multiphysics actually computes failure index, safety factor, damage index, and margin of safety at bottom, middle, and top surfaces of the shell, as well as the most critical of the three values.

Results and Discussion

The computed stresses are shown in Table 4, while Table 5 to Table 8 show the analytical values for failure index and safety factor (reserve factor) for certain failure criteria. For the Tsai-Wu Orthotropic, Modified Tsai-Hill, and Hoffman criteria, the failure index and safety factor are taken from Ref. 1. The results are compared with results from COMSOL Multiphysics.

TABLE 9: COMPARISON OF STRESSES FOR A LAYERED SHELL

Ply	σ_{11} from benchmark	σ_{11} from COMSOL	σ_{22} from benchmark	σ_{22} from COMSOL	σ_{12} from benchmark	σ_{12} from COMSOL
Ply 1	-5.128E6	-5.128E6	4.407E6	4.407E6	-1.663E6	-1.663E6
Ply 2	1.259E7	1.259E7	1.983E6	1.983E6	2.572E6	2.571E6
Ply 3	8.520E6	8.520E6	1.256E5	1.256E5	-2.051E6	-2.051E6
Ply 4	9.357E6	9.357E6	-1.859E6	-1.859E6	-5.557E5	-5.557E5

TABLE 10: COMPARISON OF FAILURE INDEX (FI) AND SAFETY FACTORS (SF) FOR PLY 1 (90 DEGREE PLY).

Criterion	FI from benchmark or analytical computations	FI from COMSOL	SF from benchmark or analytical computations	SF from COMSOL
Tsai-Wu Orthotropic	0.8840	0.8841	1.122	1.1223
Tsai-Hill	0.7795	0.7794	1.132	1.1327
Hoffman	0.8811	0.8814	1.1253	1.1258
Modified Tsai-Hill	0.7795	0.7794	1.1325	1.1327

TABLE 10: COMPARISON OF FAILURE INDEX (FI) AND SAFETY FACTORS (SF) FOR PLY 1 (90 DEGREE PLY).

Criterion	**FI from benchmark or analytical computations**	**FI from COMSOL**	**SF from benchmark or analytical computations**	**SF from COMSOL**
Azzi-Tsai-Hill	0.7796	0.7794	1.132	1.1327
Norris	0.7923	0.7883	1.126	1.1262
Tsai-Wu Anisotropic	0.8840	0.8841	1.122	1.1223

.

TABLE 11: COMPARISON OF FAILURE INDEX (FI) AND SAFETY FACTORS (SF) FOR PLY 2 (-45 DEGREE PLY).

Criterion	**FI from benchmark or analytical computations**	**FI from COMSOL**	**SF from benchmark or analytical computations**	**SF from COMSOL**
Tsai-Wu Orthotropic	0.3730	0.3731	2.5367	2.5367
Tsai-Hill	0.1632	0.1632	2.474	2.4748
Hoffman	0.3763	0.3760	2.4944	2.4941
Modified Tsai-Hill	0.1632	0.1632	2.4748	2.4748
Azzi-Tsai-Hill	0.1632	0.1632	2.474	2.4748
Norris	0.1533	0.1533	2.553	2.5534
Tsai-Wu Anisotropic	0.37308	0.3731	2.536	2.5367

TABLE 12: COMPARISON OF FAILURE INDEX (FI) AND SAFETY FACTORS (SF) FOR PLY 3(45 DEGREE PLY).

Criterion	**FI from benchmark or analytical computations**	**FI from COMSOL**	**SF from benchmark or analytical computations**	**SF from COMSOL**
Tsai-Wu Orthotropic	0.0199	0.0199	14.302	14.302
Tsai-Hill	0.0043	0.0043	15.15	15.157
Hoffman	0.0200	0.0200	14.098	14.098
Modified Tsai-Hill	0.0043	0.0043	15.157	15.157
Azzi-Tsai-Hill	0.0043	0.0043	15.15	15.157
Norris	0.0039	0.0039	15.95	15.954
Tsai-Wu Anisotropic	0.0199	0.0199	14.30	14.302

TABLE 13: COMPARISON OF FAILURE INDEX (FI) AND SAFETY FACTORS (SF) FOR PLY 4 (0 DEGREE PLY).

Criterion	**FI from benchmark or analytical computations**	**FI from COMSOL**	**SF from benchmark or analytical computations**	**SF from COMSOL**
Tsai-Wu Orthotropic	-0.3430	-0.3430	31.885	31.884
Tsai-Hill	0.1390	0.1390	2.68	2.682
Hoffman	-0.3451	-0.3450	37.876	37.876
Modified Tsai-Hill	0.00140	0.00135	27.12	27.124
Azzi-Tsai-Hill	0.00128	0.00126	27.87	27.877
Norris	0.00168	0.00168	24.38	24.388
Tsai-Wu Anisotropic	-0.3430	-0.3430	31.88	31.884

For many industrial and real life applications, the safety factor (SF) is more useful than the failure index (FI). The safety factor (or reserve factor) gives a direct indication of how close the component is to failure. Figure 2 shows the Hoffman safety factor (SF) at the mid-plane for the different plies. Ply 1 (90 degree ply) is close to failure as expected because of its orientation, where fibers are perpendicular to the loading direction.

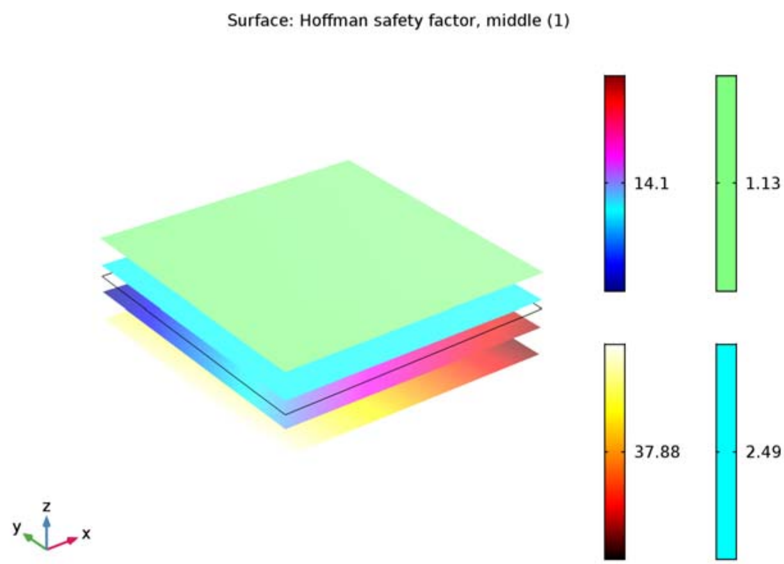


Figure 2: Hoffman safety factors at mid-planes for a stack of shells.

The von Mises stresses in all plies are shown in Figure 3. The stress in ply 1 is the lowest, but still this layer is still more susceptible to failure due to the orientation of its fibers.

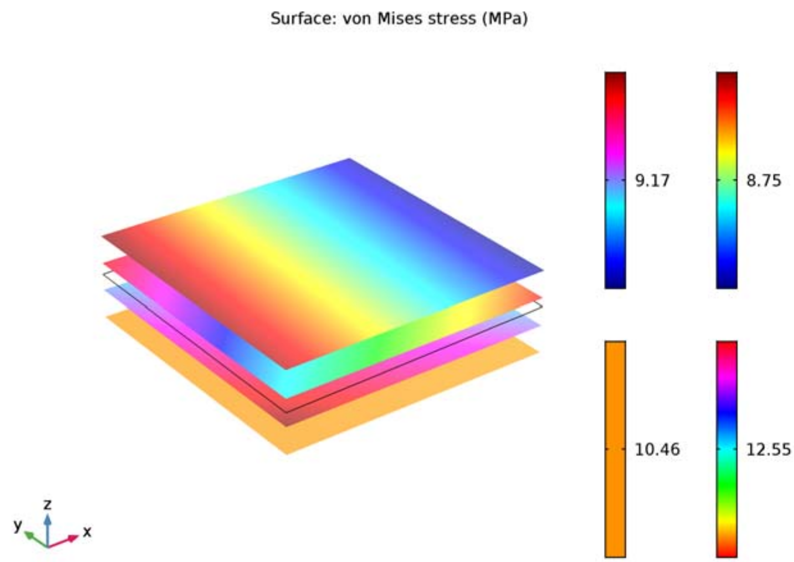


Figure 3: von Mises stress in a stack of shells.

Notes About the COMSOL Implementation

This layered shell is modeled using four separate Shell interfaces on top of each other. All four interfaces are located on the same boundary, and share the translational and rotational degrees of freedom. Is is only the different values of the offset properties which describes the stacking.

The boundary conditions provided in the benchmark specifications apply to the layered shell as a single entity. When implemented in this model, special attention must be paid to the boundary condition stating that in one point, only the x-translation should be constrained. In the shell sense, this is a condition on the mid surface of the stack, which is between ply 2 and ply 3. Setting the degree of freedom u to zero, would in this case imply that also the rotation around the y-axis is constrained, since it would be applied on all layers. The intended boundary condition is instead implemented by stating that the x-displacement in ply 3 should be the negative of the x-displacement in ply 2.

Reference

1. P. Hopkins, *Benchmarks for Membrane and Bending Analysis of Laminated Shells, Part 2: Strength Analysis,* NAFEMS, 2005.

Application Library path: `Structural_Materials_Module/Verification_Examples/failure_prediction_in_a_layered_shell`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.
7 Click **Add**.
8 In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.
9 Click **Add**.
10 Click **Study**.
11 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
12 Click **Done**.

ROOT

Parameters

On the **Home** toolbar, click **Parameters**.

GLOBAL DEFINITIONS

Parameters

Select the material strengths from Table 2.

1 In the **Settings** window for **Parameters**, locate the **Parameters** section.

2 In the table, enter the following settings:

Name	Expression	Value	Description
th	0.05e-3[m]	5E-5 m	Shell thickness
Ftotal	15[N]	15 N	Total edge load
Sigmats1	500[MPa]	5E8 Pa	Tensile strength 11
Sigmats2	5[MPa]	5E6 Pa	Tensile strength 22
Sigmats3	5[MPa]	5E6 Pa	Tensile strength 33
Sigmacs1	350[MPa]	3.5E8 Pa	Compressive strength 11
Sigmacs2	75[MPa]	7.5E7 Pa	Compressive strength 22
Sigmacs3	75[MPa]	7.5E7 Pa	Compressive strength 33
Sigmass23	35[MPa]	3.5E7 Pa	Shear strength 23
Sigmass13	35[MPa]	3.5E7 Pa	Shear strength 13
Sigmass12	35[MPa]	3.5E7 Pa	Shear strength 12

DEFINITIONS

Set up three rotated coordinate systems.

Rotated System 2 (sys2)

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Rotated System**.

2 In the **Settings** window for **Rotated System**, locate the **Settings** section.

3 Find the **Euler angles (Z-X-Z)** subsection. In the α text field, type `pi/2`.

Rotated System 3 (sys3)

1 Right-click **Rotated System 2 (sys2)** and choose **Duplicate**.

2 In the **Settings** window for **Rotated System**, locate the **Settings** section.

3 Find the **Euler angles (Z-X-Z)** subsection. In the α text field, type `-pi/4`.

Rotated System 4 (sys4)

1 Right-click **Component 1 (comp1)>Definitions>Rotated System 3 (sys3)** and choose **Duplicate**.

2 In the **Settings** window for **Rotated System**, locate the **Settings** section.

3 Find the **Euler angles (Z-X-Z)** subsection. In the α text field, type `pi/4`.

GEOMETRY 1

Plane Geometry

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)** click **Plane Geometry**.

Square 1 (sq1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `1e-2`.

4 Click **Build Selected**.

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

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

SHELL (SHELL)

Activate **Discretization** and **Advanced Physics** options from **Show** button.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

2 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

The layered shell is modeled using four separate shell interfaces located on the same boundary (mesh surface), sharing the degrees of freedom. Stacking of the shells is done using a **Physical Offset** option. With this option the constraints and loads are transfered to the actual midplane of the shells without modeling it.

As the same degrees of freedom are to be shared by all shell interfaces, set the displacement field to **u** and the displacement of shell normals to **ar** for Shell 2, Shell 3 and Shell 4.

Set the discretization for the displacement field to **Linear** in order to resemble the benchmark example.

The results given in the benchmark example are at mid-plane of each shell layer. Set the **Default Through Thickness Result Location** to zero for all shells.

3 In the **Model Builder** window, click **Shell (shell)**.

4 In the **Settings** window for **Shell**, type `Ply 1` in the **Label** text field.

5 In the **Name** text field, type `shell1`.

6 Locate the **Thickness** section. In the d text field, type `th`.

7 From the **Offset definition** list, choose **Physical offset**.

8 In the z_{offset} text field, type `1.5*th`.

9 Click to expand the **Default through-thickness result location** section. Locate the **Default Through-Thickness Result Location** section. In the z text field, type `0`.

10 Click to expand the **Discretization** section. From the **Displacement field** list, choose **Linear**.

PLY 1 (SHELL1)

On the **Physics** toolbar, click **Shell (shell)** and choose **Ply 1 (shell1)**.

Linear Elastic Material 1

Choose orthotropic solid model for linear elastic material and assign **Rotated System 2** as **Shell Local System**.

1 In the **Model Builder** window, under **Component 1 (comp1)>Ply 1 (shell1)** click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.

3 From the **Solid model** list, choose **Orthotropic**.

Shell Local System 1

1 In the **Model Builder** window, expand the **Linear Elastic Material 1** node, then click **Shell Local System 1**.

2 In the **Settings** window for **Shell Local System**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

SHELL 2 (SHELL2)

On the **Physics** toolbar, click **Ply 1 (shell1)** and choose **Shell 2 (shell2)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell 2 (shell2)**.

2 In the **Settings** window for **Shell**, type `Ply 2` in the **Label** text field.

3 Locate the **Thickness** section. In the d text field, type `th`.

4 From the **Offset definition** list, choose **Physical offset**.

5 In the z_{offset} text field, type `0.5*th`.

6 Locate the **Discretization** section. From the **Displacement field** list, choose **Linear**.

7 Locate the **Default Through-Thickness Result Location** section. In the z text field, type `0`.

8 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section. In the **Displacement field** text field, type `u`.

9 In the **Displacement of shell normals** text field, type `ar`.

PLY 2 (SHELL2)

On the **Physics** toolbar, click **Shell 2 (shell2)** and choose **Ply 2 (shell2)**.

Linear Elastic Material 1

Choose orthotropic solid model for linear elastic material and assign **Rotated System 3** as **Shell Local System**.

1 In the **Model Builder** window, under **Component 1 (comp1)>Ply 2 (shell2)** click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.

3 From the **Solid model** list, choose **Orthotropic**.

Shell Local System 1

1 In the **Model Builder** window, expand the **Linear Elastic Material 1** node, then click **Shell Local System 1**.

2 In the **Settings** window for **Shell Local System**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Rotated System 3 (sys3)**.

SHELL 3 (SHELL3)

On the **Physics** toolbar, click **Ply 2 (shell2)** and choose **Shell 3 (shell3)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell 3 (shell3)**.

2 In the **Settings** window for **Shell**, type `Ply 3` in the **Label** text field.

3 Locate the **Thickness** section. In the d text field, type `th`.

4 From the **Offset definition** list, choose **Physical offset**.

5 In the z_{offset} text field, type `-0.5*th`.

6 Locate the **Discretization** section. From the **Displacement field** list, choose **Linear**.

7 Locate the **Default Through-Thickness Result Location** section. In the z text field, type `0`.

8 Locate the **Dependent Variables** section. In the **Displacement field** text field, type `u`.

9 In the **Displacement of shell normals** text field, type `ar`.

PLY 3 (SHELL3)

On the **Physics** toolbar, click **Shell 3 (shell3)** and choose **Ply 3 (shell3)**.

Linear Elastic Material 1

Choose orthotropic solid model for linear elastic material and assign **Rotated System 4** as **Shell Local System**.

1 In the **Model Builder** window, under **Component 1 (comp1)>Ply 3 (shell3)** click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.

3 From the **Solid model** list, choose **Orthotropic**.

Shell Local System 1

1 In the **Model Builder** window, expand the **Linear Elastic Material 1** node, then click **Shell Local System 1**.

2 In the **Settings** window for **Shell Local System**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Rotated System 4 (sys4)**.

SHELL 4 (SHELL4)

On the **Physics** toolbar, click **Ply 3 (shell3)** and choose **Shell 4 (shell4)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell 4 (shell4)**.

2 In the **Settings** window for **Shell**, type `Ply 4` in the **Label** text field.

3 Locate the **Thickness** section. In the d text field, type `th`.

4 From the **Offset definition** list, choose **Physical offset**.

5 In the z_{offset} text field, type `-1.5*th`.

6 Locate the **Discretization** section. From the **Displacement field** list, choose **Linear**.

7 Locate the **Default Through-Thickness Result Location** section. In the z text field, type `0`.

8 Locate the **Dependent Variables** section. In the **Displacement field** text field, type `u`.

9 In the **Displacement of shell normals** text field, type `ar`.

PLY 4 (SHELL4)

On the **Physics** toolbar, click **Shell 4 (shell4)** and choose **Ply 4 (shell4)**.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Ply 4 (shell4)** click **Linear Elastic Material 1**.
2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.
3 From the **Solid model** list, choose **Orthotropic**.

MATERIALS

Material 1 (mat1)

Select the material properties for orthotropic material from Table 1.

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 1 (mat1)**.
2 In the **Settings** window for **Material**, locate the **Material Contents** section.
3 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Young's modulus	Evector	{207e9, 7.6e9, 7.6e9}	Pa	Orthotropic
Poisson's ratio	nuvector	{0.3,0, 0}	1	Orthotropic
Shear modulus	Gvector	{5e9, 5e9, 5e9}	N/m²	Orthotropic
Density	rho	7800	kg/m³	Basic

PLY 1 (SHELL1)

On the **Physics** toolbar, click **Ply 4 (shell4)** and choose **Ply 1 (shell1)**.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Ply 1 (shell1)** click **Linear Elastic Material 1**.

Safety 1

1 On the **Physics** toolbar, click **Attributes** and choose **Safety**.
2 In the **Settings** window for **Safety**, locate the **Failure Model** section.

3 From the **Failure criterion** list, choose **Tsai-Wu Orthotropic**.

Safety 2, 3, 4, 5, 6, 7

1 Create six similar **Safety** nodes by duplicating the above node, and replace the failure criterion as given in the table below:

Name	Failure Criterion
Safety 2	`Tsai-Hill Orthotropic`
Safety 3	`Hoffman Orthotropic`
Safety 4	`Modified Tsai-Hill Orthotropic`
Safety 5	`Azzi-Tsai-Hill Orthotropic`
Safety 6	`Norris Orthotropic`
Safety 7	`Tsai-Wu Anisotropic`

Select all **Safety** nodes under **Play 1 (shell1)>> Linear Elastic Material 1**, and right click to **Copy**. Then go to **Linear Elastic Material 1** under **Play 2 (shell2)**, **Play 3 (shell3)** and **Ply 4 (shell4)**; and right click to **Paste Mutiple Items**.

MATERIALS

Material 1 (mat1)

Enter the material properties for Tsai-Wu Anisotropic criterion as shown in Equation 1 and Equation 2.

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 1 (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Tensile strengths	sigmats	`{Sigmats1, Sigmats2, Sigmats3}`	Pa	Orthotropic strength parameters, Voigt notation
Compressive strengths	sigmacs	`{Sigmacs1, Sigmacs2, Sigmacs3}`	Pa	Orthotropic strength parameters, Voigt notation

Property	Name	Value	Unit	Property group
Shear strengths	sigmass	{Sigmass23, Sigmass13, Sigmass12}	Pa	Orthotropic strength parameters, Voigt notation
Second rank tensor, Voigt notation	F_s	{1/Sigmats1-1/Sigmacs1,1/Sigmats2-1/Sigmacs2,1/Sigmats3-1/Sigmacs3,0,0,0}	1/Pa	Anisotropic strength parameters, Voigt notation
Fourth rank tensor, Voigt notation	F_f	{1/(Sigmats1*Sigmacs1),-0.5*sqrt(1/((Sigmats1*Sigmacs1)*(Sigmats2*Sigmacs2))),1/(Sigmats2*Sigmacs2),-0.5*sqrt(1/((Sigmats1*Sigmacs1)*(Sigmats3*Sigmacs3))),-0.5*sqrt(1/((Sigmats2*Sigmacs2)*(Sigmats3*Sigmacs3))),1/(Sigmats3*Sigmacs3),0,0,0,1/Sigmass23^2,0,0,0,0,1/Sigmass13^2,0,0,0,0,0,1/Sigmass12^2}	$m^2 \cdot s^4/kg^2$	Anisotropic strength parameters, Voigt notation
Density	rho	7800	kg/m^3	Basic
Young's modulus	Evector	{207e9,7.6e9,7.6e9}	Pa	Orthotropic
Poisson's ratio	nuvector	{0.3,0,0}	1	Orthotropic
Shear modulus	Gvector	{5e9,5e9,5e9}	N/m^2	Orthotropic

Property	Name	Value	Unit	Property group
Loss factor for orthotropic Young's modulus	eta_Evector	{0,0,0}	1	Orthotropic
Loss factor for orthotropic shear modulus	eta_Gvector	{0,0,0}	1	Orthotropic

PLY 1 (SHELL1)

Fixed Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 Select Point 1 only.

Apply a nodal tensile load of 15 N as an edge load. The load is shared by all shell midplanes, hence it is divided by 4 in order to keep a total value of 15 N.

Edge Load 1

1 On the **Physics** toolbar, click **Edges** and choose **Edge Load**.

2 Select Edge 4 only.

3 In the **Settings** window for **Edge Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{\mathrm{tot}}$ vector as

Ftotal/4	x
0	y
0	z

Now select **Fixed Constraint** and **Edge Load** nodes under **Ply 1 (shell1)**, and right click to **Copy**. Then go to **Ply 2 (shell2)**, **Ply 3 (shell3)** and **Ply 4 (shell4)**; and right click to **Paste Mutiple Items**.

PLY 2 (SHELL2)

To enforce fixed x-direction translation on node 2, apply displacement u0 in x-direction for point 2 for shell 2 and displacement -u0 in x direction for same point for shell 3. Also add a **Global Equation** under shell 3 for this additional degree of freedom u0.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Ply 2 (shell2)**.

Prescribed Displacement/Rotation 1

1 On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.

2 Select Point 2 only.

3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 In the u_0 text field, type `u0`.

PLY 3 (SHELL3)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Ply 3 (shell3)**.

2 On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.

3 Select Point 2 only.

4 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.

5 Select the **Prescribed in x direction** check box.

6 In the u_0 text field, type `-u0`.

7 In the **Model Builder** window's toolbar, click the **Show** button and clear **Advanced Physics Options** in the menu.

Global Equations 1

1 In the **Model Builder** window, right-click **Ply 3 (shell3)** and choose **Global>Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt, t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
u0		0	0	

4 Locate the **Units** section. Find the **Dependent variable quantity** subsection. From the list, choose **Dispersed phase volume fraction (1)**.

5 From the list, choose **Displacement field (m)**.

MESH 1

Use a single quadrilateral element.

Free Quad 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Quad**.
2. Click in the **Graphics** window and then press Ctrl+A to select all boundaries.

Distribution 1

1. Right-click **Component 1 (comp1)>Mesh 1>Free Quad 1** and choose **Distribution**.
2. In the **Settings** window for **Distribution**, locate the **Edge Selection** section.
3. From the **Selection** list, choose **All edges**.
4. Locate the **Distribution** section. In the **Number of elements** text field, type `1`.
5. Click **Build All**.

STUDY 1

Switch off the generation of default plots, since each Shell interface will generate three plots by default.

1. In the **Settings** window for **Study**, locate the **Study Settings** section.
2. Clear the **Generate default plots** check box.
3. On the **Home** toolbar, click **Compute**.

RESULTS

In the **Model Builder** window, expand the **Results** node.

Cut Point 3D 1

1. On the **Results** toolbar, click **Cut Point 3D**.
2. In the **Settings** window for **Cut Point 3D**, locate the **Point Data** section.
3. In the **X** text field, type `0.5e-2`.
4. In the **Y** text field, type `0.5e-2`.
5. In the **Z** text field, type `0`.

Point Evaluation 1

1. On the **Results** toolbar, click **Point Evaluation**.
2. In the **Settings** window for **Point Evaluation**, type `Failure indexes in Ply 1` in the **Label** text field.
3. In the **Settings** window for **Point Evaluation**, locate the **Data** section.
4. From the **Data set** list, choose **Cut Point 3D 1**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`shell1.emm1.sf1.f_im`	1	
`shell1.emm1.sf2.f_im`	1	
`shell1.emm1.sf3.f_im`	1	
`shell1.emm1.sf4.f_im`	1	
`shell1.emm1.sf5.f_im`	1	
`shell1.emm1.sf6.f_im`	1	
`shell1.emm1.sf7.f_im`	1	

6 Click **Evaluate**.

Point Evaluation 2, 3, 4

Create three similar **Point Evaluation** nodes by duplicating the above node, and replace the word shell1 in the **Expressions** by shell2, shell3, and shell4 for **Point Evaluation 2**, **Point Evaluation 3**, and **Point Evaluation 4**, respectively. Rename them appropriately.

Point Evaluation 5

1 On the **Results** toolbar, click **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, type `Safety factors in Ply 1` in the **Label** text field.

3 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

4 From the **Data set** list, choose **Cut Point 3D 1**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`shell1.emm1.sf1.s_fm`	1	
`shell1.emm1.sf2.s_fm`	1	
`shell1.emm1.sf3.s_fm`	1	
`shell1.emm1.sf4.s_fm`	1	
`shell1.emm1.sf5.s_fm`	1	
`shell1.emm1.sf6.s_fm`	1	
`shell1.emm1.sf7.s_fm`	1	

6 Click **Evaluate**.

Point Evaluation 6, 7, 8

Create three similar **Point Evaluation** nodes by duplicating the above node and replace the word shell1 in the **Expressions** by shell2, shell3, and shell4 for **Point Evaluation 6**, **Point Evaluation 7**, and **Point Evaluation 8**, respectively. Rename them appropriately.

Point Evaluation 9

1 On the **Results** toolbar, click **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, type `Stresses in Ply 1` in the **Label** text field.

3 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

4 From the **Data set** list, choose **Cut Point 3D 1**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`shell1.Sl11`	`N/m^2`	
`shell1.Sl22`	`N/m^2`	
`shell1.Sl12`	`N/m^2`	

6 Click **Evaluate**.

Point Evaluation 10, 11, 12

Create three similar **Point Evaluation** nodes by duplicating the above node, and replace the word shell1 in the **Expressions** by shell2, shell3, and shell4 for **Point Evaluation 10**, **Point Evaluation 11**, and **Point Evaluation 12**, respectively. Rename them appropriately.

To visualize von Mises stress in the layered shell, use four different **Surface** plots for four shells in the **3D Plot Group**. Modify the Z component in the **Deformation** node for each surface in order to visualize it better.

3D Plot Group 1

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `von-Mises stress in stack of shells` in the **Label** text field.

Surface 1

1 Right-click **von-Mises stress in stack of shells** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `shell1.mises`.

4 From the **Unit** list, choose **MPa**.

Deformation 1

1 Right-click **Results>von-Mises stress in stack of shells>Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **Z component** text field, type `w+1.5e-3`.

4 Locate the **Scale** section. Select the **Scale factor** check box.

5 In the associated text field, type `1`.

Surface 2, 3, 4

Create three similar **Surface** nodes by duplicating the above node, and replace the word shell1 in the **Expression** by shell2, shell3, and shell4 for **Surface 2**, **Surface 3**, and **Surface 4**, respectively. Replace the choice of color table in the subsequent **Surface** nodes, and also replace the Z component field in the corresponding **Deformation** node with the following choices in the table:

Name	**Choice of color table**	**Z component field expression**
Surface 2	`Cyclic`	`w+0.5e-3`
Surface 3	`Disco`	`w-0.5e-3`
Surface 4	`Thermal`	`w-1.5e-3`

von-Mises stress in stack of shells

1 In the **Model Builder** window, under **Results** click **von-Mises stress in stack of shells**.

2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

3 From the **Position** list, choose **Right double**.

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

To visualize Hoffman safety factors in the layered shell, use four different **Surface** plots for four shells in the **3D Plot Group**. Modify the Z component in the **Deformation** node for each surface in order to visualize it better.

3D Plot Group 2

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Hoffman safety factors in stack of shells` in the **Label** text field.

Surface 1

1 Right-click **Hoffman safety factors in stack of shells** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `shell1.emm1.sf3.s_fm`.

Deformation 1

1 Right-click **Results>Hoffman safety factors in stack of shells>Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **Z component** text field, type `w+1.5e-3`.

4 Locate the **Scale** section. Select the **Scale factor** check box.

5 In the associated text field, type `1`.

Surface 2, 3, 4

Create three similar **Surface** nodes by duplicating the above node, and replace the word shell1 in the **Expression** by shell2, shell3, and shell4 for **Surface 2**, **Surface 3**, and **Surface 4**, respectively. Replace the choice of color table in the subsequent **Surface** nodes, and also replace the Z component field in the corresponding **Deformation** node with the following choices in the table:

Name	Choice of color table	Z component field expression
Surface 2	`Cyclic`	`w+0.5e-3`
Surface 3	`Disco`	`w-0.5e-3`
Surface 4	`Thermal`	`w-1.5e-3`

Hoffman safety factors in stack of shells

1 In the **Model Builder** window, under **Results** click **Hoffman safety factors in stack of shells**.

2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

3 From the **Position** list, choose **Right double**.

Eigenfrequency Analysis of a Free Cylinder

Introduction

In the following example you build and solve an axisymmetric model using the Solid Mechanics interface.

The model calculates the eigenfrequencies and mode shapes of an axisymmetric free cylinder. It 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*, vol. 3 (Ref. 1). The Benchmark tests the capability to handle rigid body modes and eigenfrequencies.

The cylinder is 10 m tall with an inner radius of 1.8 m and a thickness of 0.4 m.

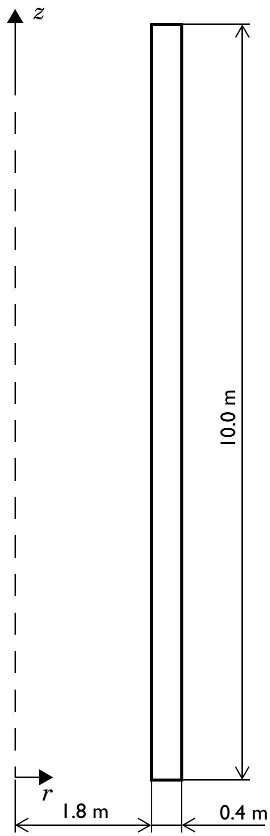


Figure 1: Model geometry in the rz-plane.

MATERIAL

Isotropic material with $E = 2.0 \cdot 10^{11}$ Pa and $\nu = 0.3$.

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 corresponding shape is a pure axial rigid body translation without any radial displacement. The eigenfrequencies are in close agreement with the target values from the NAFEMS Free Vibration Benchmarks (Ref. 1); see below.

EIGENFREQUENCY	COMSOL	TARGET (Ref. 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	394.23 Hz	394.11 Hz
f_5	397.86 Hz	397.72 Hz
f_6	405.55 Hz	405.28 Hz

Figure 2 shows the shape of the second eigenmode.

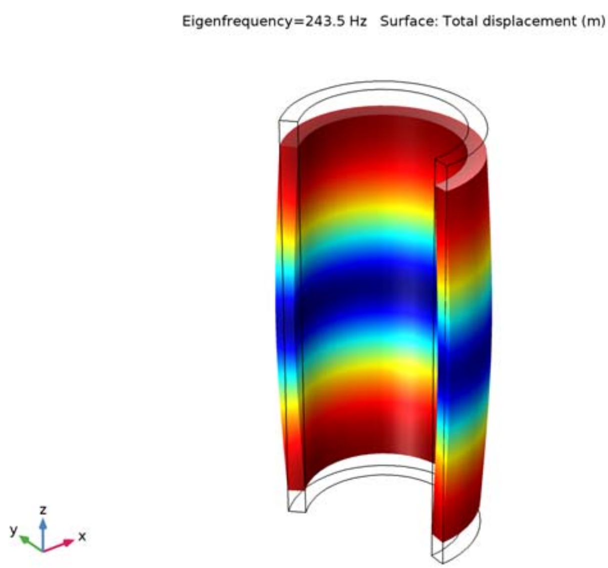


Figure 2: The second eigenmode of the cylinder.

Reference

1. F. Abassian, D.J. Dawswell, and N.C. Knowles, *Free Vibration Benchmarks, vol.3*, NAFEMS, Glasgow, 1987.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/free_cylinder`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

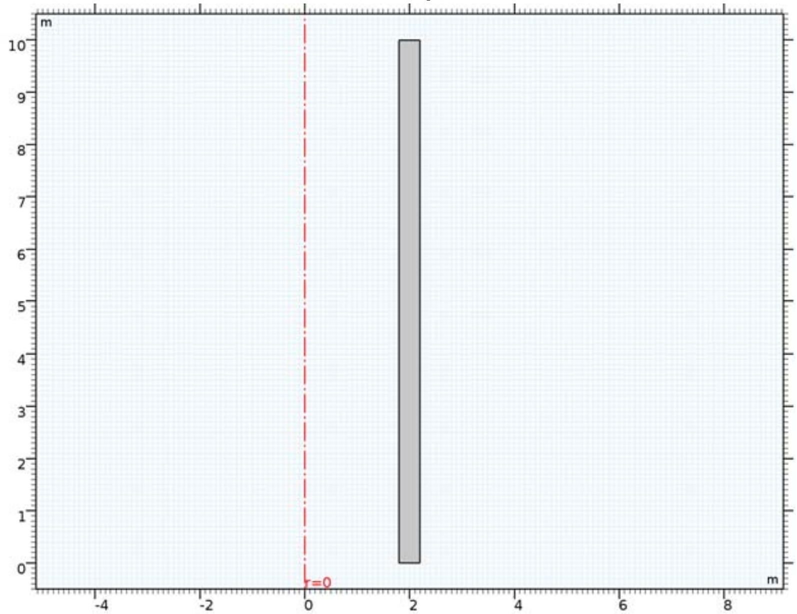
1 In the **Model Wizard** window, click **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Eigenfrequency**.
6 Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `0.4`.
4 In the **Height** text field, type `10`.
5 Locate the **Position** section. In the **r** text field, type `1.8`.
6 Click **Build All Objects**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.



SOLID MECHANICS (SOLID)

Linear Elastic Material 1

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node, then click **Linear Elastic Material 1**.
2. In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.
3. From the E list, choose **User defined**. In the associated text field, type `2e11[Pa]`.
4. From the ν list, choose **User defined**. In the associated text field, type `0.3`.
5. From the ρ list, choose **User defined**. In the associated text field, type `8000[kg/m^3]`.

STUDY 1

Step 1: Eigenfrequency

1. In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Eigenfrequency**.
2. In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
3. In the **Search for eigenfrequencies around** text field, type `100`.
4. On the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (solid)

Visualize an eigenmode in 3D.

Mode Shape, 3D (solid)

1 In the **Model Builder** window, under **Results** click **Mode Shape, 3D (solid)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Eigenfrequency (Hz)** list, choose **243.5**.
4 Locate the **Color Legend** section. Clear the **Show legends** check box.
5 Click the **Show Grid** button on the **Graphics** toolbar.
6 On the **Mode Shape, 3D (solid)** toolbar, click **Plot**.
7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Created in COMSOL Multiphysics 5.3

In-Plane and Space Truss

Introduction

In the following example you first build and solve a simple 2D truss model using the 2D Truss interface. Later on, you analyze a 3D variant of the same problem using the 3D Truss interface. This model calculates the deformation and forces 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 2D geometry consists of a square symmetrical truss built up by five members. All members have the same cross-sectional area. The side length is L, and the Young's modulus is E.

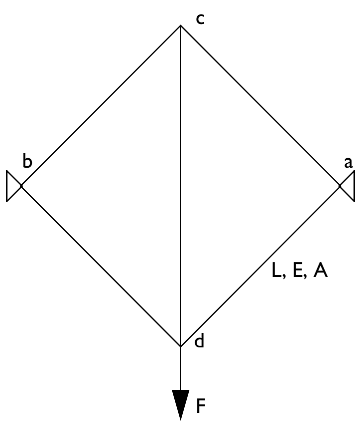


Figure 1: The truss geometry.

In the 3D case, another copy of the diagonal bars are rotated 90° around the vertical axis so that a cube with one space diagonal is generated. The figure above is thus applicable to a view in the zy-plane as well as in the xy-plane. The central bar is then given the twice the area of the other members. In this way, a space truss with exactly the same type of symmetry, but twice the vertical stiffness is generated.

GEOMETRY

- Truss side length, L = 2 m
- The truss members have a circular cross section with a radius of 0.05 m. In the 3D case, the area of the central bar is doubled.

MATERIAL

Aluminum: Young's modulus, E = 70 GPa.

CONSTRAINTS

Displacements in both directions are constrained at a and b. In the 3D the two new points are constrained in the same way.

LOAD

A vertical force F of 50 kN is applied at the bottom corner. In the 3D case, the value 100 kN is used instead in order to get the same displacements.

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	$-5.14\cdot10^{-4}$ m	$-5.15\cdot10^{-4}$ m
Displacement at c	$-2.13\cdot10^{-4}$ m	$-2.13\cdot10^{-4}$ 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 nearly perfect agreement.

Figure 2 and Figure 3 show plots visualizing the deformed geometry together with the axial forces in the truss members.

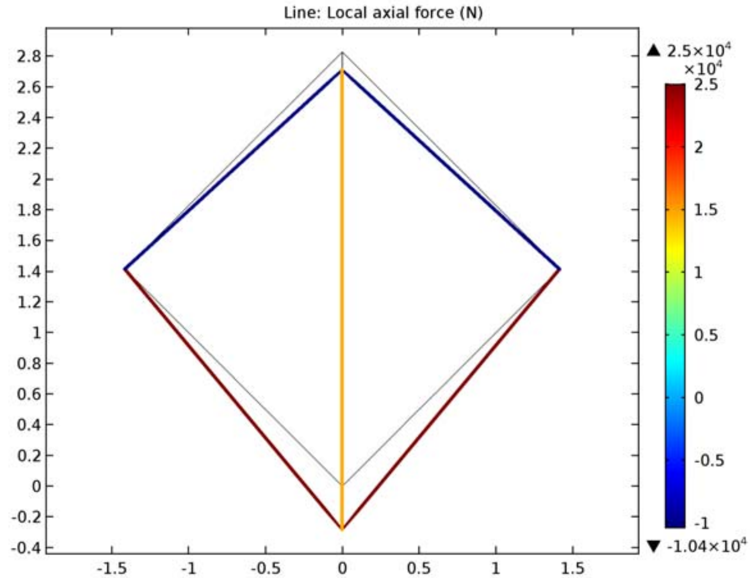


Figure 2: Deformed geometry and axial forces for the 2D case.

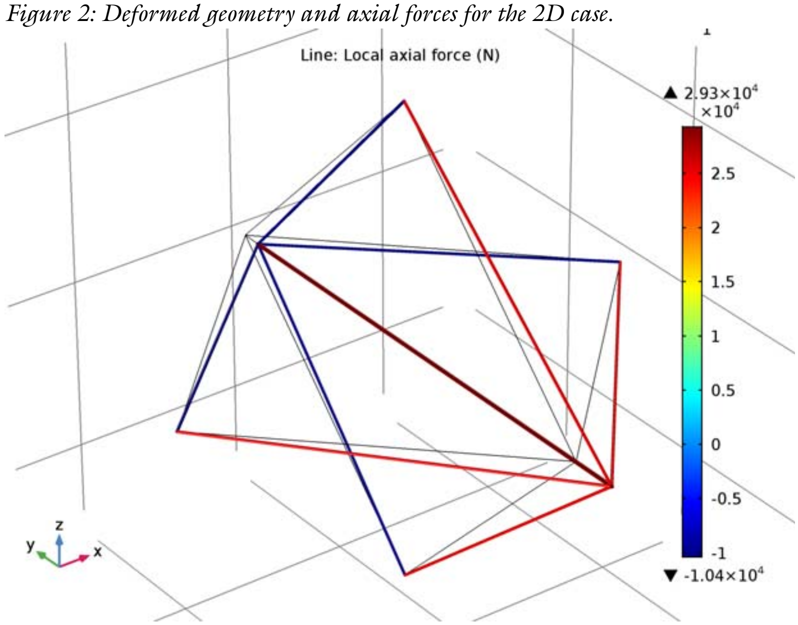


Figure 3: Deformed geometry and axial forces for the 3D case.

Notes About the COMSOL Implementation

In this example you build the 2D and the 3D truss as two different components within the same MPH file. This is not essential, you could equally well choose to create the components in separate MPH files.

Reference

1. T.H.G. Megson, *Aircraft Structures for Engineering Students*, Edward Arnold, p. 404, 1985

Application Library path: `Structural_Mechanics_Module/Verification_Examples/inplane_and_space_truss`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, click **2D**.
2. In the **Select Physics** tree, select **Structural Mechanics>Truss (truss)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

GEOMETRY 1

Square 1 (sq1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Square**.
2. In the **Settings** window for Square, locate the **Size** section.
3. In the **Side length** text field, type 2.
4. Locate the **Rotation Angle** section. In the **Rotation** text field, type 45.

5 Locate the **Object Type** section. From the **Type** list, choose **Curve**.

6 Click **Build All Objects**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **y** to `sqrt(8)`.

5 Click **Build All Objects**.

TRUSS (TRUSS)

Cross Section Data 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Truss (truss)** click **Cross Section Data 1**.

2 In the **Settings** window for Cross Section Data, locate the **Cross Section Data** section.

3 In the A text field, type `pi/4*0.05^2`.

Pinned 1

1 On the **Physics** toolbar, click **Points** and choose **Pinned**.

2 Select Points 1 and 4 only.

Point Load 1

1 On the **Physics** toolbar, click **Points** and choose **Point Load**.

2 Select Point 2 only.

3 In the **Settings** window for Point Load, locate the **Force** section.

4 Specify the $\mathbf{F}_\mathrm{P}$ vector as

0	x
-50e3	y

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	70e9	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	2900	kg/m³	Basic

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Force (truss)

1. In the **Model Builder** window, click **Force (truss)**.
2. In the **Settings** window for 2D Plot Group, click to expand the **Color legend** section.
3. Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.
4. Click the **Zoom Extents** button on the **Graphics** toolbar.

Derived Values

Next, compute the displacements at d (Vertex 2) and c (Vertex 3).

Point Evaluation 1

1. On the **Results** toolbar, click **Point Evaluation**.
2. Select Points 2 and 3 only.
3. In the **Settings** window for Point Evaluation, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Truss>Displacement>Displacement field>v - Displacement field, y component**.
4. Click **Evaluate**.

   Although you can read off the values of the local axial force in the members ac and ad from the max and min values for the color legend for the plot in the Graphics window, it is instructive to see how you can compute such values more generally.

DEFINITIONS

Add average component coupling operators for the members ac, ad, and cd. You will use these for defining variables that evaluate to the axial forces in these members.

Average 1 (aveop1)

1. On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for Average, type `aveop_ac` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 5 only.

Average 2 (aveop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for Average, type `aveop_ad` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.

Average 3 (aveop3)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for Average, type `aveop_cd` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 3 only.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for Variables, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
F_ac	aveop_ac(truss.Nxl)	N	Axial force, member ac
F_ad	aveop_ad(truss.Nxl)	N	Axial force, member ad
F_cd	aveop_cd(truss.Nxl)	N	Axial force, member cd

STUDY 1

Update the solution to evaluate the variables you just defined.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

2 Right-click **Solution 1 (sol1)** and choose **Solution>Update**.

RESULTS

Global Evaluation 1

1. On the **Results** toolbar, click **Global Evaluation**.
2. In the **Settings** window for Global Evaluation, locate the **Expressions** section.
3. In the table, enter the following settings:

Expression	Unit	Description
F_ac	N	Axial force, member ac
F_ad	N	Axial force, member ad
F_cd	N	Axial force, member cd

4. Click **Evaluate**.

   The values in the Table window agree with those of the analytical reference solution.

   Now create the 3D truss as a new model.

ROOT

On the **Home** toolbar, click **Add Component** and choose **3D**.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

ADD PHYSICS

1. On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2. Go to the **Add Physics** window.
3. In the tree, select **Recently Used>Truss (truss)**.
4. Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.
5. Click **Add to Component** in the window toolbar.
6. On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1. On the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

   Switch off the 2D truss physics in this study.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Truss (truss)** interface.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

GEOMETRY 2

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for Work Plane, click **Show Work Plane**.

Square 1 (sq1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for Square, locate the **Size** section.

3 In the **Side length** text field, type `2`.

4 Locate the **Rotation Angle** section. In the **Rotation** text field, type `45`.

5 Locate the **Object Type** section. From the **Type** list, choose **Curve**.

6 On the **Work Plane** toolbar, click **Build All**.

7 In the **Model Builder** window, click **Geometry 2**.

Rotate 1 (rot1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 In the **Settings** window for Rotate, locate the **Input** section.

3 Select the **Keep input objects** check box.

4 Select the object **wp1** only.

5 Locate the **Axis of Rotation** section. From the **Axis type** list, choose **Cartesian**.

6 In the **y** text field, type `1`.

7 In the **z** text field, type `0`.

8 Locate the **Rotation Angle** section. In the **Rotation** text field, type `90`.

9 Click **Build All Objects**.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **y** to `sqrt(8)`.

5 Click **Build All Objects**.

DEFINITIONS

Add average component coupling operators for the members ac, ad, and cd and corresponding axial force variables.

Average 4 (aveop4)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for Average, type `aveop_ac` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Edge**.

4 Select Edge 8 only.

Average 5 (aveop5)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for Average, type `aveop_ad` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Edge**.

4 Select Edge 4 only.

Average 6 (aveop6)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for Average, type `aveop_cd` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Edge**.

4 Select Edge 5 only.

Variables 2

1 On the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for Variables, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
F_ac	aveop_ac(truss2.Nxl)	N	Axial force, member ac
F_ad	aveop_ad(truss2.Nxl)	N	Axial force, member ad
F_cd	aveop_cd(truss2.Nxl)	N	Axial force, member cd

TRUSS 2 (TRUSS2)

Cross Section Data 1

1 In the **Model Builder** window, expand the **Component 2 (comp2)>Truss 2 (truss2)** node, then click **Cross Section Data 1**.
2 In the **Settings** window for Cross Section Data, locate the **Cross Section Data** section.
3 In the A text field, type `pi/4*0.05^2`.

Cross Section Data 2

1 On the **Physics** toolbar, click **Edges** and choose **Cross Section Data**.
2 Select Edge 5 only.
3 In the **Settings** window for Cross Section Data, locate the **Cross Section Data** section.
4 In the A text field, type `2*pi/4*0.05^2`.

Pinned 1

1 On the **Physics** toolbar, click **Points** and choose **Pinned**.
2 Select Points 1, 3, 4, and 6 only.

Point Load 1

1 On the **Physics** toolbar, click **Points** and choose **Point Load**.
2 Select Point 2 only.
3 In the **Settings** window for Point Load, locate the **Force** section.
4 Specify the $\mathbf{F}_P$ vector as

0	x
-100e3	y
0	z

MATERIALS

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **Blank Material**.

Material 2 (mat2)

1 In the **Settings** window for Material, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	70e9	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	2900	kg/m³	Basic

STUDY 2

On the **Home** toolbar, click **Compute**.

RESULTS

Force (truss2)

1 In the **Model Builder** window, under **Results** click **Force (truss2)**.

2 In the **Settings** window for 3D Plot Group, click to expand the **Color legend** section.

3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Derived Values

Proceed to compute the displacements at d (Vertex 2) and c (Vertex 5).

1 In the **Model Builder** window, under **Results** click **Derived Values**.

Point Evaluation 2

1 On the **Results** toolbar, click **Point Evaluation**.

2 In the **Settings** window for Point Evaluation, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 Select Points 2 and 5 only.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 2>Truss 2>Displacement>Displacement field>v2 - Displacement field, y component**.

6 Click **New Table**.

TABLE

1 Go to the **Table** window.

The results are nearly identical to those of the 2D case.

Finally, compute the axial force values.

RESULTS

Global Evaluation 2

1. On the **Results** toolbar, click **Global Evaluation**.
2. In the **Settings** window for Global Evaluation, locate the **Data** section.
3. From the **Data set** list, choose **Study 2/Solution 2 (2) (sol2)**.
4. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`comp2.F_ac`	N	Axial force, member ac
`comp2.F_ad`	N	Axial force, member ad
`comp2.F_cd/2`	N	Axial force, member cd

   Because the applied force was doubled to get the same displacement as, in the 2D case, you need to divide the value of the axial force in member cd by 2 to get a value comparable to that of the 2D case.

5. Click **Evaluate**.

   Again, the values in the Results table agree very well with the reference solution.

Created in COMSOL Multiphysics 5.3

In-Plane Framework with Discrete Mass and Mass Moment of Inertia

Introduction

In the following example you build and solve a 2D beam model using the 2D Structural Mechanics Beam interface. This example describes the eigenfrequency analysis of a simple geometry. A point mass and point mass moment of inertia are used. 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 Figure 1 below).

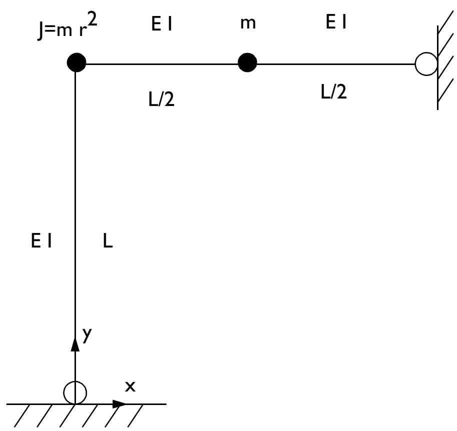


Figure 1: Definition of the problem.

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 as $L/4$.

CONSTRAINTS

The beam is pinned at $x = 0, y = 0$ and $x = 1, y = 1$, meaning that the displacements 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:

$$\omega_{e1}^2 = \frac{48EI}{mL^3}$$

$$\omega_{e2}^2 = \frac{48 \cdot 32EI}{7mL^3}$$

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

The following two plots visualize the two eigenmodes.

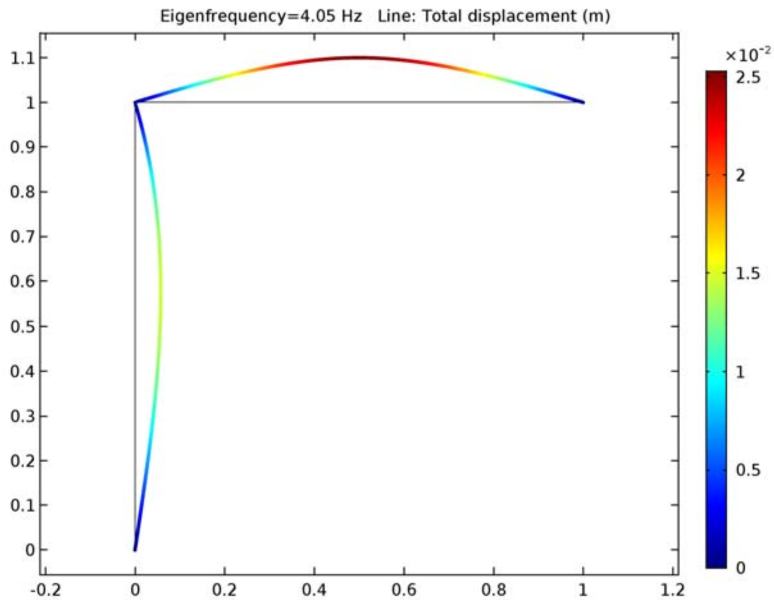


Figure 2: The first eigenmode.

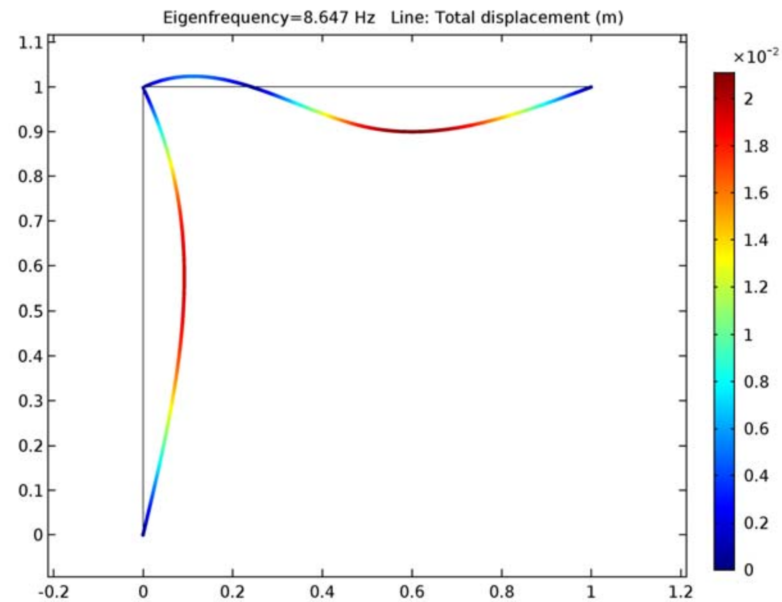


Figure 3: The second eigenmode.

Because the beams have no density in this example, the total mass is the 1000 kg supplied by the point mass. The mass represented by the computed eigenmodes can be evaluated using the mass participation factors, see Figure 4. In this case, it can be seen that in the y direction, the correspondence is perfect, while almost none of the mass in the x direction is represented. The axial deformation mode for the horizontal member has a higher frequency, and was not computed. Further information on participation factors is given in Structural Mechanics Module > Structural Mechanics Modeling > Eigenfrequency Analysis.

Messages | Progress | Log | Table 1

Eigenfrequency	Participation factor u	Participation factor v	MPF_comp1.v^2
4.05009	-0.00854	25.31359	640.77802
8.64738	-0.01148	-18.95315	359.22198

Figure 4: Participation factors for each eigenfrequency.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/inplane_framework_freq`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.
2 In the **Select Physics** tree, select **Structural Mechanics>Beam (beam)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Eigenfrequency**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
a	0.03[m]	0.03 m	Side length
Emod	200[GPa]	2E11 Pa	Young's modulus
I	(a)^4/12	6.75E-8 m^4	Area moment of inertia
L	1[m]	1 m	Framework member length
m	1000[kg]	1000 kg	Point mass
r	L/4	0.25 m	Point mass radius
J	m*r^2	62.5 kg·m²	Point mass moment of inertia

Name	Expression	Value	Description
A	a*a	9E-4 m²	Cross-sectional area
w1	sqrt(48*Emod*I/(m*L^3))	25.46 1/s	Angular frequency, eigenfrequency 1
w2	sqrt(48*32*Emod*I/(7*m*L^3))	54.43 1/s	Angular frequency, eigenfrequency 2
f1	w1/(2*pi)	4.051 1/s	Eigenfrequency 1
f2	w2/(2*pi)	8.662 1/s	Eigenfrequency 2

GEOMETRY 1

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for Bézier Polygon, locate the **General** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.

5 Find the **Control points** subsection. In row **2**, set **y** to L.

6 Find the **Added segments** subsection. Click **Add Linear**.

7 Find the **Control points** subsection. In row **2**, set **x** to L/2.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **x** to L.

10 Click **Build All Objects**.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	Emod	Pa	Basic
Poisson's ratio	nu	0	1	Basic
Density	rho	0	kg/m³	Basic

BEAM (BEAM)

Cross Section Data 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Beam (beam)** node, then click **Cross Section Data 1**.
2 In the **Settings** window for Cross Section Data, locate the **Cross Section Definition** section.
3 From the list, choose **Common sections**.
4 In the h_y text field, type `a`.
5 In the h_z text field, type `a`.

Pinned 1

1 On the **Physics** toolbar, click **Points** and choose **Pinned**.
2 Select Points 1 and 4 only.

Point Mass 1

1 On the **Physics** toolbar, click **Points** and choose **Point Mass**.
2 Select Point 3 only.
3 In the **Settings** window for Point Mass, locate the **Point Mass** section.
4 In the m text field, type `m`.

Point Mass 2

1 On the **Physics** toolbar, click **Points** and choose **Point Mass**.
2 Select Point 2 only.
3 In the **Settings** window for Point Mass, locate the **Point Mass** section.
4 In the J_z text field, type `J`.

STUDY 1

Step 1: Eigenfrequency

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Eigenfrequency**.
2 In the **Settings** window for Eigenfrequency, locate the **Study Settings** section.
3 Select the **Desired number of eigenfrequencies** check box.
4 In the associated text field, type `2`.

   Change the scaling of the eigenmodes in order to compute modal participation factors and modal masses.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Eigenvalue Solver 1**.

4 In the **Settings** window for Eigenvalue Solver, locate the **Output** section.

5 From the **Scaling of eigenvectors** list, choose **Mass matrix**.

6 On the **Study** toolbar, click **Compute**.

RESULTS

Line 1

1 In the **Model Builder** window, expand the **Results>Mode Shape (beam)** node, then click **Line 1**.

2 On the **Mode Shape (beam)** toolbar, click **Plot**.

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

Mode Shape (beam)

1 In the **Model Builder** window, under **Results** click **Mode Shape (beam)**.

2 In the **Settings** window for 2D Plot Group, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **8.647**.

4 On the **Mode Shape (beam)** toolbar, click **Plot**.

Examine the modal participation factors.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Solver>MPF_comp1.u - Participation factor u**.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`MPF_comp1.u`		`Participation factor u`
`MPF_comp1.v`		`Participation factor v`
`MPF_comp1.v^2`		

4 Click **New Table**.

Global Evaluation 2

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Solver>MPF_comp1.v - Participation factor v**.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression
MPF_comp1.v^2

4 Locate the **Data Series Operation** section. From the **Operation** list, choose **Integral**.

5 Click **Evaluate**.

Kirsch Infinite Plate Problem

Introduction

In this example, you perform a static stress analysis to obtain the stress distribution in the vicinity of a small hole in an infinite plate. Two approximations of the infinite plate are evaluated. The first one uses a plate that is large compared to the hole while the second one employs an infinite element domain.

The problem is a classic benchmark, and the theoretical solution was derived by G. Kirsch in 1898. This implementation is based on the Kirsch plate model described on page 184 in *Mechanics of Materials,* D. Roylance (Ref. 1). The stress level is compared with the theoretical values.

Model Definition

Model the infinite plate in a 2D plane stress approximation 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 you need to analyze only a quarter of the plate, see Figure 1. Choose the size of the plate sufficiently large so that the stress concentration close to the hole is not affected.

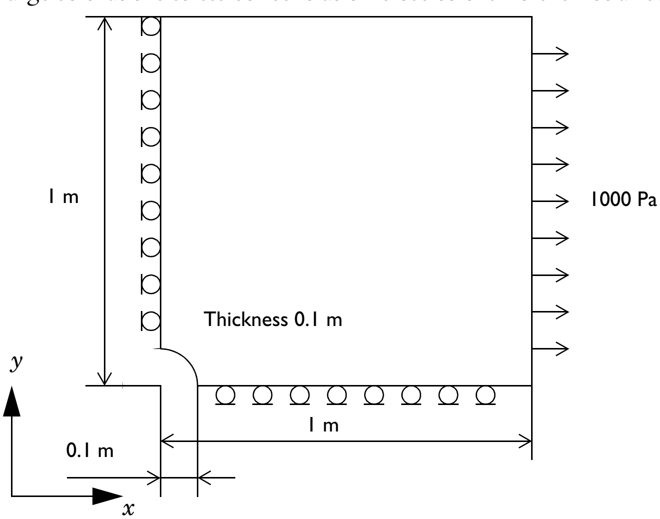


Figure 1: Geometry model of the Kirsch plate with rollers defining the symmetry plane.

When modeling a plate using the infinite element domain you need to create an additional layers around the plate. Those layers simulate the part that stretches to infinity and can have an arbitrarily length along the direction that stretches to infinity, for example 0.1 m.

In the model the infinite element domain is created along the y direction only since the numerical results along $x = 0$ symmetry plane are compared to an analytical reference and infinite element domain in x direction only have a minor influence.

MATERIAL

Isotropic material with, $E = 2.1 \cdot 10^{11}$ Pa, $\nu = 0.3$.

LOAD

A distributed stress of 10^3 Pa on the right edge pointing in the x direction.

CONSTRAINTS

Symmetry planes, $x = 0$, $y = 0$.

Results

The distribution of the normal stress in the x-direction, σ_x, is shown in Figure 2. The stress contours of both the finite model and infinite model are very similar.

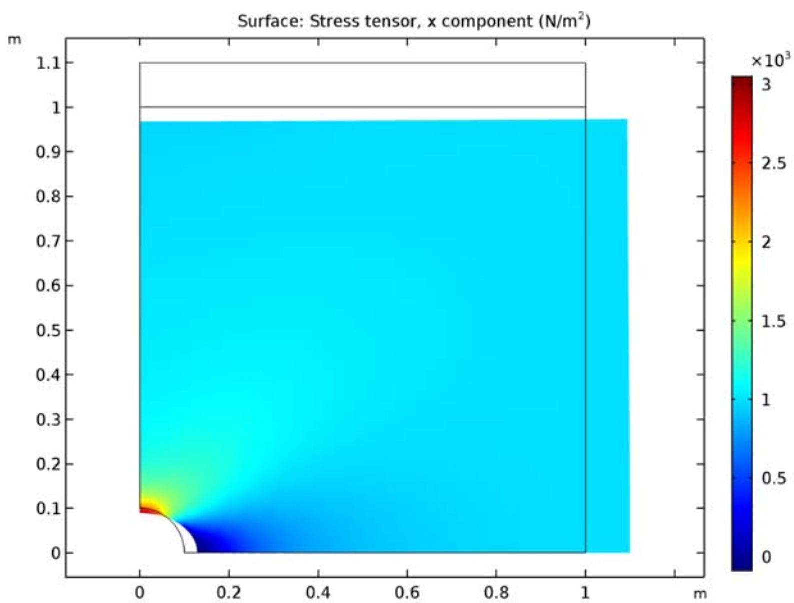


Figure 2: Distribution of the normal stress in the x direction for finite model.

According to Ref. 1 the stress σ_x along the vertical symmetry line can be calculated as

$$\sigma_x = \frac{1000}{2}\left(2 + \frac{0{,}1^2}{y^2} + 3\frac{0{,}1^4}{y^4}\right) \qquad (1)$$

Figure 3 shows the stress σ_x obtained from the solved models, and plotted as a function of the y-coordinate along the left symmetry edge, which are in close agreement with the theoretical value according to Equation 1.

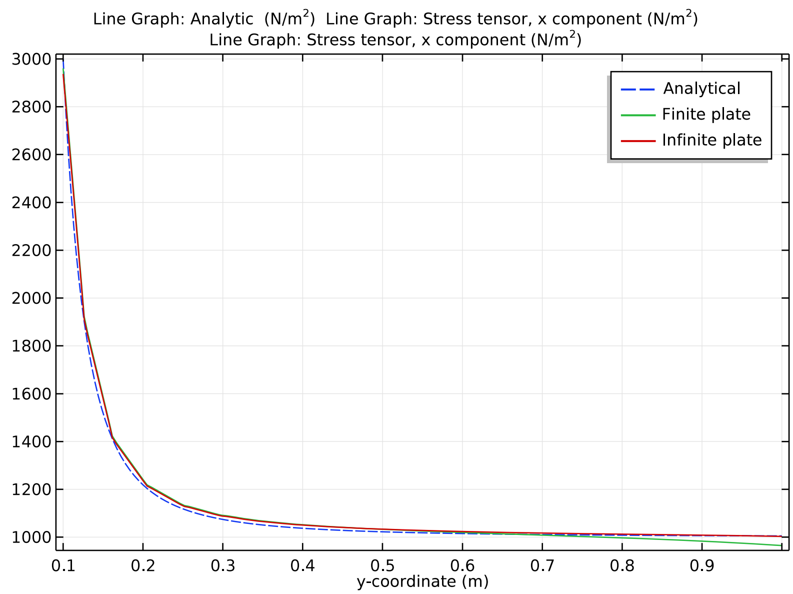


Figure 3: Normal stress, simulated results (solid line) versus the theoretical values (dashed line).

Away from the hole, stresses from the finite model starts drifting from the theoretical values, while stresses from the infinite model matches closely with theoretical value.

Notes About the COMSOL Implementation

The infinite element domain gives best results when meshed with rectangular elements, see Figure 4.

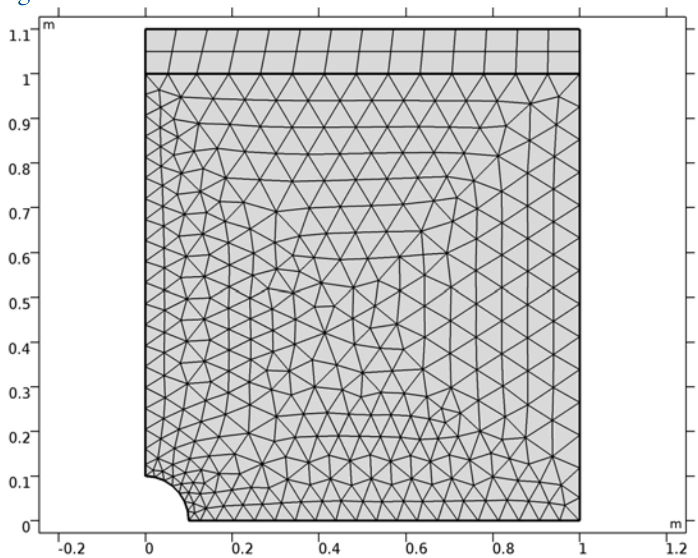


Figure 4: Infinite element domain modeled with rectangular elements.

Reference

1. D. Roylance, *Mechanics of Materials*, John Wiley & Sons, 1996.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/kirsch_plate`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

Square 1 (sq1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.

2 In the **Model Builder** window, right-click **Square 1 (sq1)** and choose **Build Selected**.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.1`.

4 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Add the square and remove the circle in the **Difference** section.

3 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

4 In the **Settings** window for **Difference**, locate the **Selections of Resulting Entities** section.

5 Select the **Resulting objects selection** check box.

6 In the **Label** text field, type `Finite Plate`.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Height** text field, type `0.1`.

4 Locate the **Position** section. In the **y** text field, type `1`.

First add an analytical function for stress, based on Kirsch's theoretical solution of an infinite plate.

Analytic 1 (an1)

On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

GLOBAL DEFINITIONS

Analytic 1 (an1)

1 In the **Settings** window for **Analytic**, type `Analytic` in the **Label** text field.

2 In the **Function name** text field, type `AnaStress`.

3 Locate the **Definition** section. In the **Expression** text field, type `1000/2*(2+0.1^2/y^2+3*0.1^4/y^4)`.

4 In the **Arguments** text field, type `y`.

5 Locate the **Units** section. In the **Arguments** text field, type `m`.

6 In the **Function** text field, type `N/m^2`.

SOLID MECHANICS (SOLID)

First set up a model without the **Infinite Element Domain**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 In the list, select **2**.

4 Click **Remove from Selection**.

5 Select Domain 1 only.

6 Locate the **2D Approximation** section. From the list, choose **Plane stress**.

7 Locate the **Thickness** section. In the d text field, type `0.1`.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1, 2, and 5 only.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundaries 6 and 7 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

1e3	x
0	y

Now set up a model with the **Infinite Element Domain**.

ADD PHYSICS

1 On the **Physics** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.

4 Click **Add to Component** in the window toolbar.

5 On the **Physics** toolbar, click **Add Physics** to close the **Add Physics** window.

SOLID MECHANICS 2 (SOLID2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 2 (solid2)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.

3 From the list, choose **Plane stress**.

4 Locate the **Thickness** section. In the d text field, type `0.1`.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1, 2, and 5 only.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundaries 6 and 7 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

1e3	x
0	y

DEFINITIONS

Infinite Element Domain 1 (ie1)

1 On the **Definitions** toolbar, click **Infinite Element Domain**.

2 Select Domain 2 only.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	2.1e11	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	7800	kg/m³	Basic

MESH 1

For the **Finite Plate** selection, a customized **Free Triangular** mesh must be used for getting a better solution in the stress concentration region. A customized triangular mesh is created by choosing different number of elements along the periphery of the hole and symmetry edges of the **Finite Plate**.

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Finite Plate**.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Distribution**.

2 Select Boundary 8 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 7.

Distribution 2

1 Right-click **Free Triangular 1** and choose **Distribution**.

2 Select Boundaries 1 and 5 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `20`.

5 Click **Build All**.

Infinite Element Domain gives best results when meshed with rectangular elements.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 Click **Build All**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

To check the error in the computed results, make a point evaluation of stresses near the hole ($y = 0.1$) and away from the hole ($y = 1$) for the solution computed with and without **Infinite Element Domain**. The error can be determined by finding the difference between computed stresses and analytical stresses.

Point Evaluation 1

1 On the **Results** toolbar, click **Point Evaluation**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`(solid.SXX-AnaStress(0.1))/AnaStress(0.1)`	1	

5 In the **Label** text field, type `Error Evaluation for Finite Plate`.

6 Click **Evaluate**.

Point Evaluation 2

1 On the **Results** toolbar, click **Point Evaluation**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Evaluation**, type `Error Evaluation for Finite Plate 1` in the **Label** text field.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`(solid.SXX-AnaStress(1))/AnaStress(1)`	1	

5 Click **Table 1 - Error Evaluation for Finite Plate ((solid.SXX-AnaStress(0.1))/AnaStress(0.1))**.

Point Evaluation 3

1 On the **Results** toolbar, click **Point Evaluation**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Evaluation**, type `Error Evaluation for Infinite Plate` in the **Label** text field.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
(solid2.SXX-AnaStress(0.1))/AnaStress(0.1)	1	

5 Click **Table 1 - Error Evaluation for Finite Plate ((solid.SXX-AnaStress(0.1))/AnaStress(0.1))**.

Point Evaluation 4

1 On the **Results** toolbar, click **Point Evaluation**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
(solid2.SXX-AnaStress(1))/AnaStress(1)	1	

5 In the **Label** text field, type `Error Evaluation for Infinite Plate 1`.

6 Click **Table 1 - Error Evaluation for Finite Plate ((solid.SXX-AnaStress(0.1))/AnaStress(0.1))**.

Stress (solid)

The default plot shows the von Mises stress combined with a scaled deformation of the plate. Display the stress field in the x-direction instead since the external load is oriented in that direction.

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Stress>Stress tensor (spatial frame)>solid.sx - Stress tensor, x component**.

3 On the **Stress (solid)** toolbar, click **Plot**.

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

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress (solid2)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics 2>Stress>Stress tensor (spatial frame)>solid2.sx - Stress tensor, x component**.

Line Graph 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Model Builder** window, right-click **1D Plot Group 3** and choose **Line Graph**.

3 Select Boundary 1 only.

4 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

5 In the **Expression** text field, type `AnaStress(y)`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `y`.

8 On the **1D Plot Group 3** toolbar, click **Plot**.

9 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

10 Select the **Show legends** check box.

11 In the table, enter the following settings:

Legends
`Analytical`

12 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

13 On the **1D Plot Group 3** toolbar, click **Plot**.

Line Graph 2

1 Right-click **1D Plot Group 3** and choose **Line Graph**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Stress>Stress tensor (spatial frame)>solid.sx - Stress tensor, x component**.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `y`.

6 Locate the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Finite plate`

Line Graph 3

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 3** and choose **Line Graph**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics 2>Stress>Stress tensor (spatial frame)>solid2.sx - Stress tensor, x component**.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `y`.

6 Locate the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Infinite plate`

9 On the **1D Plot Group 3** toolbar, click **Plot**.

Large Deformation Analysis of a 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). A schematic description of the beam and its characteristics is shown in Figure 1.

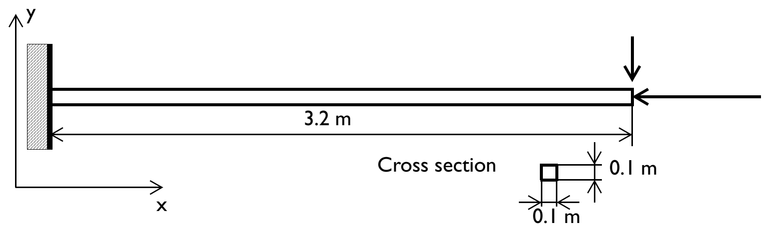


Figure 1: Cantilever beam geometry.

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 $E = 2.1 \cdot 10^{11}\ \mathrm{N/m^2}$ and $\nu = 0$.

CONSTRAINTS AND LOADS

- The left end is fixed.
- The right end is subjected to total load of $F_x = -3.844 \cdot 10^6$ N and $F_y = -3.844 \cdot 10^3$ N.

MODELING IN COMSOL

This problem is modeled separately using both Solid Mechanics and Beam Interfaces and results are compared with the Benchmark value. In Solid mechanics interface, problem is modeled as ‘plane stress’ problem considering that out-of-plane dimension is small. Poisson’s ratio ν is set to zero to make the boundary conditions consistent with the beam theory assumptions. Load on the right end of the beam is modeled as uniformly distributed boundary load corresponding to the specified total load.

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 purpose. An alternative approach would be to introduce an initial imperfection in the geometry.

Figure 2 below shows the final state with the **1:1** displacement scaling.

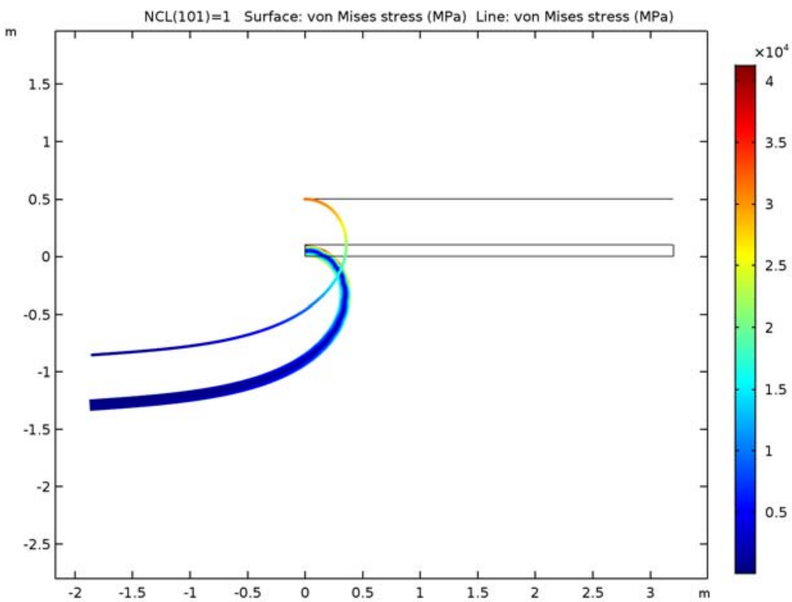


Figure 2: The effective von Mises stress of the deformed beam.

The horizontal and vertical displacements of the tip versus the compressive load normalized by its maximum value are shown in Figure 3.

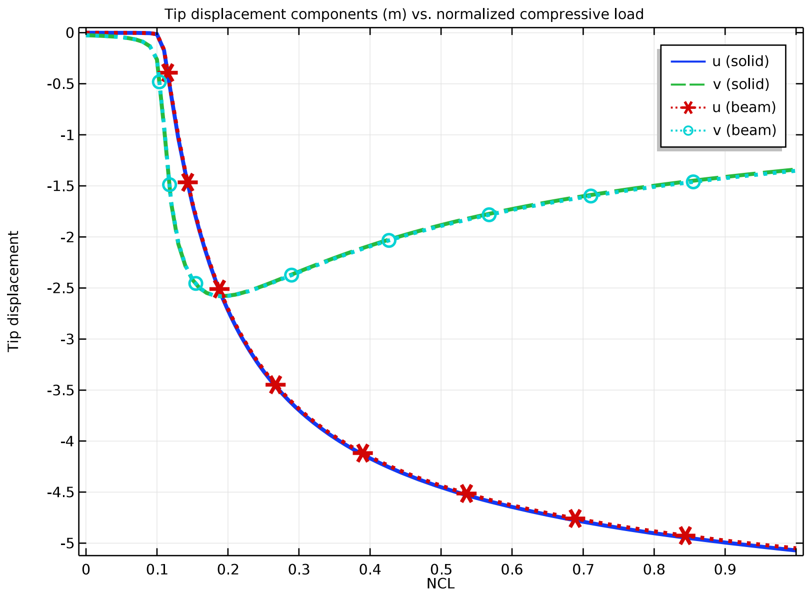


Figure 3: Horizontal and vertical tip displacements versus normalized compressive load.

Table 1 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:

TABLE 1: COMPARISON BETWEEN MODEL RESULTS AND REFERENCE VALUES.

QUANTITY	COMSOL (SOLID)	COMSOL (BEAM)	REFERENCE
Maximum vertical displacement at the tip	-2.58	-2.58	-2.58 ± 0.02
Final vertical displacement at the tip	-1.34	-1.35	-1.36 ± 0.02
Final horizontal displacement at the tip	-5.07	-5.05	-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 compressive load $3.84 \cdot 10^5$ N. It is often seen in practice that the critical load of an imperfect structure is significantly lower than that of the ideal structure.

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

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

Reference

1. A.A. Becker, *Background to Finite Element Analysis of Geometric Non-linearity Benchmarks*, NAFEMS, Ref: -R0065, Glasgow, 1999.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/large_deformation_beam`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Structural Mechanics>Beam (beam)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

8 Click **Done**.

GLOBAL DEFINITIONS

Define parameters for the geometric data, compressive and transverse load components as well as a parameter that you will use to gradually turn up the compressive load.

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
F_Lx	-3.844[MN]	-3.844E6 N	Maximum compressive load
F_Ly	1e-3*F_Lx	-3844 N	Transverse load
NCL	0	0	Normalized compressive load
d	0.1[m]	0.1 m	Cross-section dimension of the beam
l	3.2[m]	3.2 m	Length of the beam

By restricting the range for the parameter **NCL** to [0, 1], it serves as a compressive load normalized by the maximum compressive load.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `l`.

4 In the **Height** text field, type `d`.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **y** to `5*d`.

5 In row **2**, set **x** to `l` and **y** to `5*d`.

6 Locate the **General** section. From the **Type** list, choose **Open curve**.

7 Click **Build All Objects**.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

3 In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	2.1e11	Pa	Basic
Poisson's ratio	nu	0	1	Basic
Density	rho	7850	kg/m³	Basic

Material 2 (mat2)

1 In the **Model Builder** window, right-click **Material 1 (mat1)** and choose **Duplicate**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.

3 From the list, choose **Plane stress**.

4 Locate the **Thickness** section. In the d text field, type `d`.

Fixed Constraint 1

1 Right-click **Component 1 (comp1)>Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 Select Boundary 1 only.

Boundary Load 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Boundary Load**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{tot}$ vector as

NCL*F_Lx	x
F_Ly	y

BEAM (BEAM)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Beam (beam)**.

2 In the **Settings** window for **Beam**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 4 only.

Cross Section Data 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)** click **Cross Section Data 1**.

2 In the **Settings** window for **Cross Section Data**, locate the **Cross Section Definition** section.

3 From the list, choose **Common sections**.

4 In the h_y text field, type `d`.

5 In the h_z text field, type `d`.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Beam (beam)** and choose **Fixed Constraint**.

2 Select Point 3 only.

Point Load 1

1 Right-click **Beam (beam)** and choose **Point Load**.

2 In the **Settings** window for **Point Load**, locate the **Force** section.

3 Specify the $\mathbf{F}_P$ vector as

NCL*F_Lx	x
F_Ly	y

4 Select Point 6 only.

MESH 1

Edge 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Edge**.

2 Select Boundaries 2–4 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Edge 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Select Boundary 4 only.

5 Locate the **Distribution** section. In the **Number of elements** text field, type `40`.

Distribution 2

1 Right-click **Edge 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Select Boundaries 2 and 3 only.

5 Locate the **Distribution** section. In the **Number of elements** text field, type `20`.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, click **Build All**.

STUDY 1

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Study Settings** section.

2 Select the **Include geometric nonlinearity** check box.

3 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list
NCL	`range(0,0.01,1)`

6 Right-click **Study 1>Step 1: Stationary** and choose **Get Initial Value for Step**.

STUDY 1

In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)** node, then click **Stationary Solver 1**.
2 In the **Settings** window for **Stationary Solver**, locate the **General** section.
3 In the **Relative tolerance** text field, type `1e-4`.
4 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node.
5 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** and choose **Segregated**.
6 In the **Settings** window for **Segregated**, locate the **General** section.
7 From the **Termination technique** list, choose **Iterations**.
8 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1>Segregated 1** node, then click **Segregated Step**.
9 In the **Settings** window for **Segregated Step**, locate the **General** section.
10 In the **Variables** list, select **Displacement field (material and geometry frames) (comp1.beam.uLin)**.
11 Under **Variables**, click **Delete**.
12 In the **Variables** list, select **Rotation field (material and geometry frames) (comp1.beam.thLin)**.
13 Under **Variables**, click **Delete**.
14 Click to expand the **Method and termination** section. Locate the **Method and Termination** section. From the **Termination technique** list, choose **Tolerance**.
15 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Stationary Solver 1** right-click **Segregated 1** and choose **Segregated Step**.
16 In the **Settings** window for **Segregated Step**, locate the **General** section.
17 Under **Variables**, click **Add**.
18 In the **Add** dialog box, In the **Variables** list, choose **Rotation field (material and geometry frames) (comp1.beam.thLin)** and **Displacement field (material and geometry frames) (comp1.beam.uLin)**.
19 Click **OK**.
20 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.
21 From the **Nonlinear method** list, choose **Automatic (Newton)**.
22 From the **Termination criterion** list, choose **Residual**.

23 In the **Maximum number of iterations** text field, type `500`.

24 In the **Tolerance factor** text field, type `1`.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Results while solving** section.

3 Locate the **Results While Solving** section. Select the **Plot** check box.

4 From the **Plot group** list, choose **Stress (beam)**.

5 On the **Home** toolbar, click **Compute**.

RESULTS

Line 1

1 In the **Model Builder** window, expand the **Results>Stress (beam)** node, then click **Line 1**.

2 In the **Settings** window for **Line**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

4 Right-click **Results>Stress (beam)>Line 1** and choose **Copy**.

Line 1

1 In the **Model Builder** window, under **Results** right-click **Stress (solid)** and choose **Paste Line**.

2 In the **Settings** window for **Line**, type `Stress (solid and beam)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, under **Results>Stress (solid)** click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

Stress (solid and beam)

1 In the **Model Builder** window, under **Results>Stress (solid)** click **Stress (solid and beam)**.

2 In the **Settings** window for **Line**, click to expand the **Inherit style** section.

3 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

4 Clear the **Tube radius scale factor** check box.

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 On the **Stress (solid)** toolbar, click **Plot**.

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

Add a data set to use for plotting of the results at the tip of the solid beam.

Cut Point 2D 1

1 On the **Results** toolbar, click **Cut Point 2D**.

2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.

3 In the **X** text field, type `1`.

4 In the **Y** text field, type `d/2`.

5 Click **Plot**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

1D Plot Group 6

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Tip displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Point 2D 1**.

Point Graph 1

1 Right-click **Tip displacement** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Displacement>Displacement field (material and geometry frames)>u - Displacement field, X component**.

3 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `3`.

4 Click to expand the **Legends** section. Select the **Show legends** check box.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
u (solid)

Point Graph 2

1 In the **Model Builder** window, under **Results** right-click **Tip displacement** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>**

Displacement>Displacement field (material and geometry frames)>v - Displacement field, Y component.

3 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

4 In the **Width** text field, type `3`.

5 Locate the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`v (solid)`

Point Graph 3

1 Right-click **Tip displacement** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution 1 (sol1)**.

4 Locate the **Selection** section. Select the **Active** toggle button.

5 Select Point 6 only.

6 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Beam>Displacement>Displacement field (spatial frame)>u2 - Displacement field, x component**.

7 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

9 In the **Width** text field, type `3`.

10 Locate the **Legends** section. Select the **Show legends** check box.

11 From the **Legends** list, choose **Manual**.

12 In the table, enter the following settings:

Legends
`u (beam)`

Point Graph 4

1 Right-click **Results>Tip displacement>Point Graph 3** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Beam> Displacement>Displacement field (spatial frame)>v2 - Displacement field, y component**.

3 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`v (beam)`

5 On the **Tip displacement** toolbar, click **Plot**.

Tip displacement

1 In the **Model Builder** window, under **Results** click **Tip displacement**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Tip displacement components (m) vs. normalized compressive load`.

5 Locate the **Plot Settings** section. Select the **y-axis label** check box.

6 In the associated text field, type `Tip displacement`.

7 On the **Tip displacement** toolbar, click **Plot**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

Evaluate the deformation of the structure.

Point Evaluation 1

1 On the **Results** toolbar, click **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 2D 1**.

4 From the **Parameter selection (NCL)** list, choose **Last**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Solid Mechanics>Displacement> Displacement field (material and geometry frames)>u - Displacement field, X component**.

6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
u	m	Solid: x-disp
v	m	Solid: y-disp

7 Click **Evaluate**.

Point Evaluation 2

1 Right-click **Point Evaluation 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution 1 (sol1)**.

4 Locate the **Selection** section. Select the **Active** toggle button.

5 Select Point 6 only.

6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
u2	m	Beam: x-disp
v2	m	Beam: y-disp

7 Click **Table 1 - Point Evaluation 1 (u, v)**.

Created in COMSOL Multiphysics 5.3

Vibrating Beam in Fluid Flow

Introduction

A classical flow pattern is the von Kármán vortex street that can form as fluid flows past an object. These vortices may induce vibrations in the object. This problem involves a fluid-structure interaction where the large deformation affects the flow path.

The magnitude and the frequencies of the oscillation generated by the fluid around the structure are computed and compared with the values proposed by Turek and Horn; see Ref. 1.

Model Definition

The model geometry consists of a structure inside a channel with a fluid flow as represented in Figure 1 below.

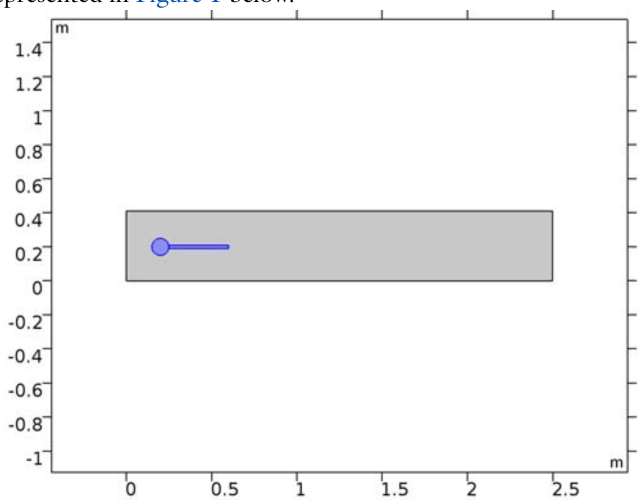


Figure 1: Model geometry including solid and fluid domains (blue and gray, respectively).

The fluid domain is a 2.5 m long and 0.41 m high channel. The structure is composed of a fixed circular domain with 0.05 m radius and centered at $(0.2, 0.2)$. The second domain of the structure is a 0.35 m by 0.02 m rectangular beam made of elastic material.

The fluid enters the channel from the left with a mean velocity of 2 m/s, and the inlet velocity profile is assumed to be fully developed.

With the inlet boundary so close to the solid structure, one can expect the inlet velocity condition to affect the flow pattern. To avoid such an effect, one might need to increase the distance between the inlet boundary and the solid structure. For the sake of comparison, the geometry in this model is kept as it is in the reference paper (Ref. 1).

The Reynolds number based on the diameter of the circle is about 200.

The fluid and solid properties are represented in table below:

TABLE 1: FLUID AND SOLID MATERIAL PROPERTIES

PARAMETER	VALUE
Fluid density	10^3 kg/m^3
Dynamic viscosity	1 Pa·s
Young's modulus	5.6 MPa
Poisson ratio	0.4

The quantities of interest are the beam rear tip displacements and the fluid forces acting on the structure. The magnitude and frequency targets (Ref. 1) are represented in the table below:

TABLE 2: TARGET RESULTS

PARAMETER	MAGNITUDE	FREQUENCY
x-displacement	-2.69±2.53 mm	10.9 Hz
y-displacement	1.48±34.38 mm	5.3 Hz
Drag	457.3±22.66 N	10.9 Hz
Lift	2.22±149.78 N	5.3 Hz

Results and Discussion

Figure 2 shows the velocity field and the von Mises stress in the structure on the deformed shape at different times. Note the von Kármán vortex street past the structure, which is significantly deformed and affects the flow field.

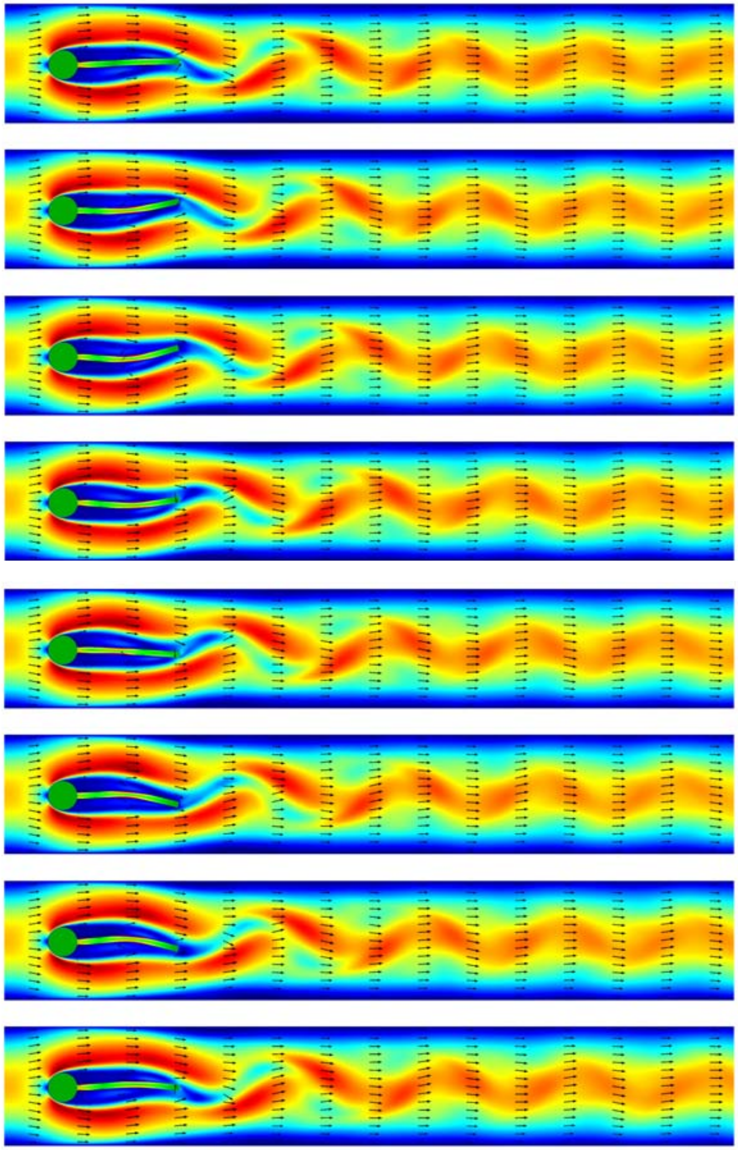


Figure 2: Velocity field in fluid and von Mises stress in structure for eight different time steps.

Figure 3 below shows the evolution of the fluid forces all along the time step. The oscillation are fully developed after $t = 3.5$ s. This is due to the external perturbation added at $t = 1.5$ s. Without this perturbation, the oscillation would develop after a longer time. Note that the oscillation can develop with some time shift due to nonlinearities in the model.

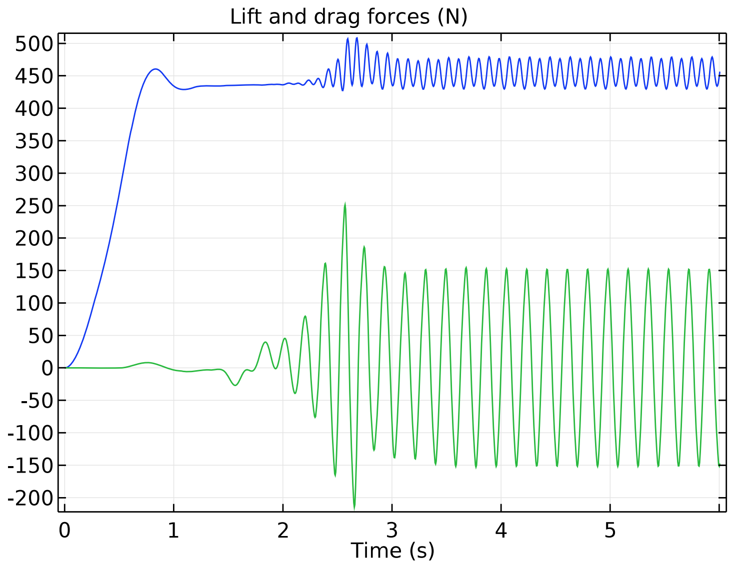


Figure 3: Drag and lift forces versus time.

Figure 4 shows the displacement of the tip of the beam in the x and y directions:

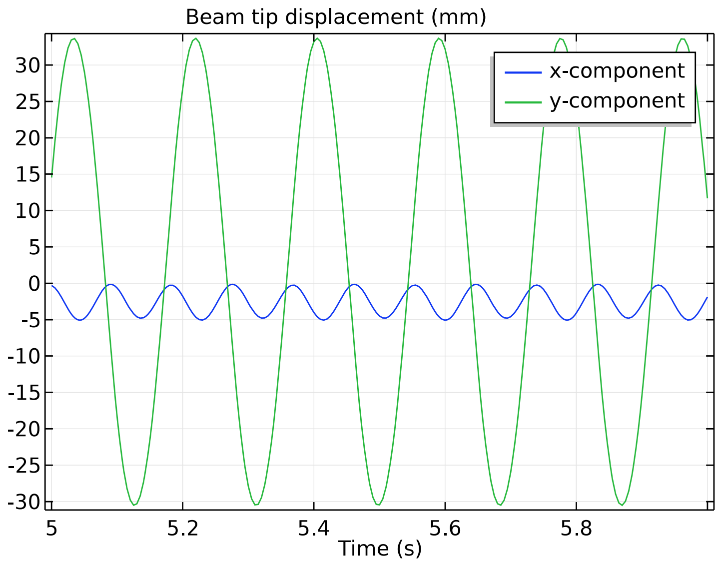


Figure 4: Tip displacement of the structure in the x and y directions (in green and blue respectively).

In the above figure, you can see that the magnitude of the x-displacement oscillation is about 2.5 mm around the average of –2.5 mm. The y-displacement varies around 2 mm with an oscillation magnitude of 32 mm, in good agreement with the targeted value.

[Figure 5](#) below shows the frequency spectrum of the structure oscillation.

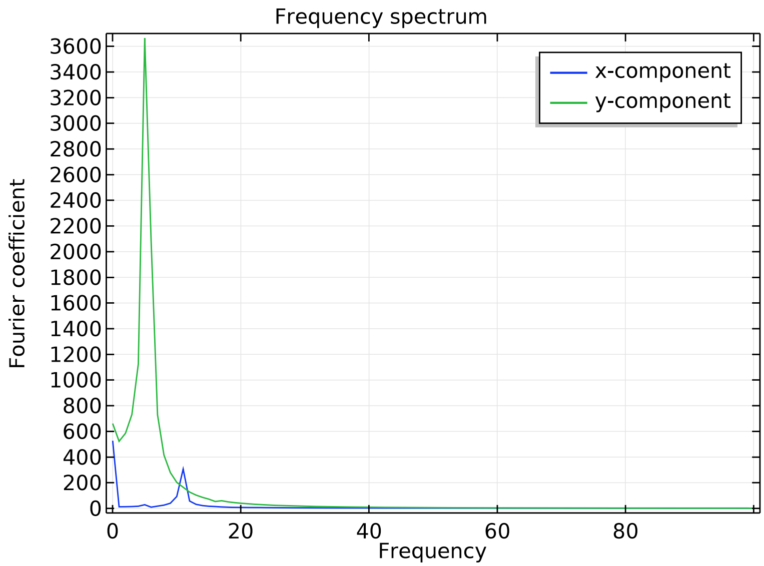


Figure 5: Frequency spectrum of the structure tip displacement.

The peaks show the main frequencies of the harmonic oscillation. For the x-displacement, the frequency is about 11 Hz, while for the y-displacement the main frequency is about 5 Hz, which agree well with the targeted results.

Figure 6 below shows the variations of the lift and drag forces applied to the structure:

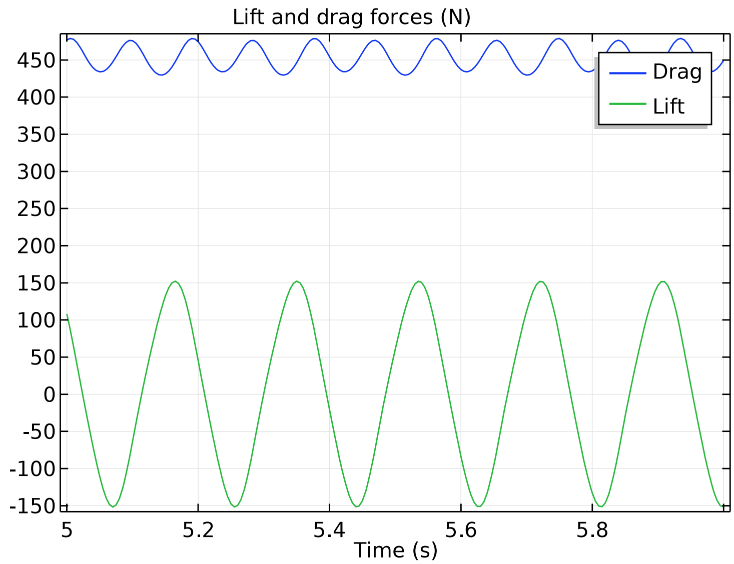


Figure 6: Lift and drag forces (green and blue curves, respectively).

The average of the total lift force is about -4 N with an oscillation magnitude of 154 N, while the drag force average is about 457 N with an oscillation magnitude of 23 N.

Notes About the COMSOL Implementation

The default discretization for the flow equations in the fluid-structure interface is based on P1+P1 elements. This means that linear order elements are used for the velocity variables. Such discretization is more stable for high Reynolds number but has lower accuracy especially in the forces evaluation. In this model, the Reynolds number is about 200, which is low enough to allow the use of P2+P1 elements for the flow equations and no consistent stabilization without expecting numerical instabilities.

Reference

1. S. Turek and J. Hron, *Proposal for numerical benchmarking of fluid-structure interaction between an elastic object and laminar incompressible flow*, Institute for Applied Mathematics and Numerics, University of Dortmund.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/oscillating_fsi`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.
2 In the **Select Physics** tree, select **Fluid Flow>Fluid-Structure Interaction (fsi)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6 Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `2.5`.
4 In the **Height** text field, type `0.41`.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Radius** text field, type `0.05`.
4 Locate the **Position** section. In the **x** text field, type `0.2`.
5 In the **y** text field, type `0.2`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.35+0.05`.

4 In the **Height** text field, type `0.02`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **x** text field, type `0.2+0.4/2`.

7 In the **y** text field, type `0.2`.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **c1** and **r2** only.

3 In the **Settings** window for **Union**, locate the **Selections of Resulting Entities** section.

4 Select the **Resulting objects selection** check box.

5 From the **Show in physics** list, choose **Domain selection**.

6 Right-click **Union 1 (uni1)** and choose **Build Selected**.

Delete Entities 1 (del1)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **uni1**, select Boundaries 1–3 only.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

2 Click the **Zoom Extents** button on the **Graphics** toolbar.

FLUID-STRUCTURE INTERACTION (FSI)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Fluid-Structure Interaction (fsi)** click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Union 1**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Union 1**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`5.6[MPa]`	Pa	Basic
Poisson's ratio	nu	`0.4`	1	Basic
Density	rho	`1e3`	kg/m³	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 Select Domain 1 only.

3 In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	`1e3`	kg/m³	Basic
Dynamic viscosity	mu	`1`	Pa·s	Basic

DEFINITIONS

Step 1 (step1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Step**.

2 In the **Settings** window for **Step**, locate the **Parameters** section.

3 In the **Location** text field, type `0.5`.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `1`.

Gaussian Pulse 1 (gp1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Gaussian Pulse**.

2 In the **Settings** window for **Gaussian Pulse**, locate the **Parameters** section.

3 In the **Location** text field, type `1.5`.

4 In the **Standard deviation** text field, type `5e-2`.

5 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

FLUID-STRUCTURE INTERACTION (FSI)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Fluid-Structure Interaction (fsi)**.
2 In the **Settings** window for **Fluid-Structure Interaction**, click to expand the **Discretization** section.
3 From the **Discretization of fluids** list, choose **P2+P1**.
4 In the **Model Builder** window's toolbar, click the **Show** button and select **Stabilization** in the menu.
5 Click to expand the **Consistent stabilization** section. Locate the **Consistent Stabilization** section. Find the **Navier-Stokes equations** subsection. Clear the **Crosswind diffusion** check box.

Fixed Constraint 1

1 Right-click **Component 1 (comp1)>Fluid-Structure Interaction (fsi)** and choose the domain setting **Solid Mechanics>Fixed Constraint**.
2 Select Domain 3 only.

Point Load 1

1 In the **Model Builder** window, right-click **Fluid-Structure Interaction (fsi)** and choose **Points>Point Load**.
2 Select Point 9 only.
3 In the **Settings** window for **Point Load**, locate the **Force** section.
4 Specify the $\mathbf{F}_p$ vector as

`0`	x
`gp1(t)`	y

Inlet 1

1 Right-click **Fluid-Structure Interaction (fsi)** and choose the boundary condition **Laminar Flow>Inlet**.
2 Select Boundary 1 only.
3 In the **Settings** window for **Inlet**, locate the **Velocity** section.
4 In the U_0 text field, type `1.5*2[m/s]*y*(0.41[m]-y)/(0.41[m]/2)^2*step1(t/1[s])`.

Outlet 1

1 Right-click **Fluid-Structure Interaction (fsi)** and choose the boundary condition **Laminar Flow>Outlet**.

2 Select Boundary 7 only.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, click **Build All**.

You can now prepare the probe variables to display during the computation.

DEFINITIONS

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 4–6 and 8–11 only.

Global Variable Probe 1 (var1)

1 On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `drag` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `-intop1(fsi.T_stressx)`.

Global Variable Probe 2 (var2)

1 On the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `lift` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `-intop1(fsi.T_stressy)`.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Times** text field, type `0 range(5,5e-3,6)`.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 1** node, then click **Displacement field (material and geometry frames) (comp1.u_solid)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `5e-2`.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Dependent Variables 1** click **Pressure (comp1.p)**.

7 In the **Settings** window for **Field**, locate the **Scaling** section.

8 From the **Method** list, choose **Manual**.

9 In the **Scale** text field, type `1e4`.

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Dependent Variables 1** click **Velocity field (spatial frame) (comp1.u_fluid)**.

11 In the **Settings** window for **Field**, locate the **Scaling** section.

12 From the **Method** list, choose **Manual**.

13 In the **Scale** text field, type `5`.

14 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Time-Dependent Solver 1**.

15 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

16 Locate the **Time Stepping** section. Select the **Maximum step** check box.

17 In the associated text field, type `5e-3`.

18 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** and choose **Fully Coupled**.

19 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.

20 Locate the **Method and Termination** section. From the **Jacobian update** list, choose **Once per time step**.

21 On the **Study** toolbar, click **Compute**.

RESULTS

Flow and Stress (fsi)

The first plot group shows the von Mises stress in the structure together with the fluid velocity magnitude.

1 In the **Model Builder** window, under **Results** click **Flow and Stress (fsi)**.

2 On the **Flow and Stress (fsi)** toolbar, click **Animation** and choose **Player**.

Probe Plot Group 3

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 3**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** check box.

4 In the associated text field, type `Time (s)`.

5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Lift and drag forces (N)`.

7 On the **Probe Plot Group 3** toolbar, click **Plot**.

Study 1/Solution 1 (3) (sol1)

1 On the **Results** toolbar, click **More Data Sets** and choose **Solution**.

2 In the **Settings** window for **Solution**, locate the **Solution** section.

3 From the **Frame** list, choose **Material (X, Y, Z)**.

Cut Point 2D 1

1 On the **Results** toolbar, click **Cut Point 2D**.

2 In the **Settings** window for **Cut Point 2D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution 1 (3) (sol1)**.

4 Locate the **Point Data** section. In the **X** text field, type `0.595`.

5 In the **Y** text field, type `0.2`.

1D Plot Group 4

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 2D 1**.

4 From the **Time selection** list, choose **Interpolated**.

5 In the **Times (s)** text field, type `range(5,5e-3,6)`.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Beam tip displacement (mm)`.

Point Graph 1

1 Right-click **1D Plot Group 4** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `u_solid`.

4 From the **Unit** list, choose **mm**.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
x-component

Point Graph 2

1 Right-click **Results>1D Plot Group 4>Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `v_solid`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
y-component

5 On the **1D Plot Group 4** toolbar, click **Plot**.

1D Plot Group 4

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, type `Beam tip displacement` in the **Label** text field.

Beam tip displacement 1

1 Right-click **Results>Beam tip displacement** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, locate the **Title** section.

3 In the **Title** text area, type `Frequency spectrum`.

Point Graph 1

1 In the **Model Builder** window, expand the **Beam tip displacement 1** node, then click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Frequency spectrum**.

Point Graph 2

1 In the **Model Builder** window, under **Results>Beam tip displacement 1** click **Point Graph 2**.

2 In the **Settings** window for **Point Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Frequency spectrum**.

4 On the **Beam tip displacement 1** toolbar, click **Plot**.

Beam tip displacement 1

1 In the **Model Builder** window, under **Results** click **Beam tip displacement 1**.

2 In the **Settings** window for **1D Plot Group**, type `Frequency spectrum` in the **Label** text field.

1D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Lift and drag forces` in the **Label** text field.

3 Locate the **Data** section. From the **Time selection** list, choose **Interpolated**.

4 In the **Times (s)** text field, type `range(5,5e-3,6)`.

5 Locate the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Lift and drag forces (N)`.

Global 1

1 Right-click **Lift and drag forces** and choose **Global**.

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>drag - Probe variable drag**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`drag`	`N/m`	`Drag`

4 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>lift - Probe variable lift**.

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`lift`	`N/m`	`Lift`

6 On the **Lift and drag forces** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.3

Pinched Hemispherical Shell

Introduction

This example studies the deformation of a hemispherical shell, where the loads cause significant geometric nonlinearity. The maximum deflections are more than two magnitudes larger than the thickness of the shell. The problem is a standard benchmark, used for testing shell formulations in a case which contains membrane and bending action, as well as large rigid body rotation. It is described in Ref. 1.

Model Definition

Figure 1 shows the geometry and the applied loads. Due to the double symmetry, the model only includes one quarter of the hemisphere.

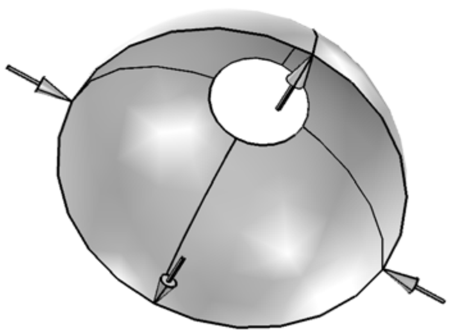


Figure 1: The geometry and loads.

The material is linear elastic with E = 68.25 MPa and ν = 0.3. The radius of the hemisphere is 10 m, and the thickness of the shell is 0.04 m. The hole at the top has a radius of 3.0902 m because 18° in the meridional direction from the top has been removed. The forces all have the value 200 N before taking symmetry into account. In the model, two forces of 100 N are applied in the symmetry planes at the lower edge of the shell.

Results and Discussion

The target solution in Ref. 1 is u = –5.952 m under the inward acting load and v = 3.427 m under the outward acting load. Both target values have an error bound of ±2%. The values computed in COMSOL are u = –5.862 m and v = 3.407 m. Both values are within 2% of the target. Figure 2 shows the deformed shape of the shell together with contours for the effective stress.

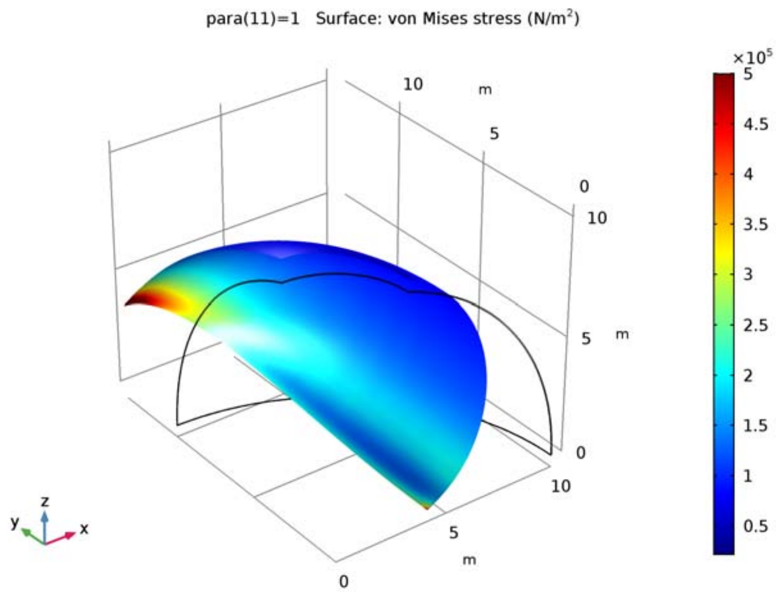


Figure 2: von Mises stress on top surface.

The change in the displacement as the load parameter increases is shown in Figure 3. As can be seen, the nonlinear effects are strong. The incremental stiffness with respect to the y-direction force increases by one order of magnitude during the loading.

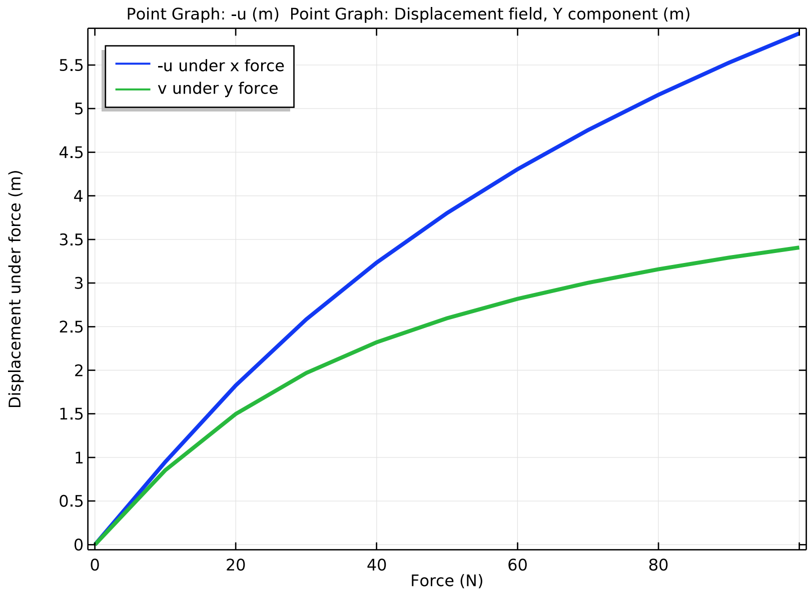


Figure 3: Displacements as functions of applied load.

Notes About the COMSOL Implementation

In a highly nonlinear problem it is a good idea to use the parametric continuation solver to track the solution instead of trying to solve at the full load. Several solver settings can be tuned to improve the convergence. Due to the large difference between the bending and the membrane stiffnesses in a thin shell, a small error in the approximated displacements during the iterations can cause large residual forces. For this reason, manual control of the damping is used in the Newton method. This will often improve solution speed for problems with severe geometrical nonlinearities.

Because the model uses point loads, the gradients are steep close to the locations where the loads are applied. For this reason you modify the distribution of the elements so that finer elements are generated toward the corners of the model. From a computational point of view, this is more effective than using a uniform refinement of the mesh.

Reference

1. N.K. Prinja and R.A. Clegg, "A Review of Benchmark Problems for Geometric Non-linear Behaviour of 3-D Beams and Shells (SUMMARY)," *NAFEMS Ref: R0024*, pp. F9A–F9B, 1993.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/pinched_hemispherical_shell`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `10`.

4 Right-click **Sphere 1 (sph1)** and choose **Build Selected**.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `10`.

4 In the **Depth** text field, type `10`.

5 In the **Height** text field, type `10`.

6 Locate the **Position** section. In the **x** text field, type `-5`.

7 In the **y** text field, type `-5`.

8 In the **z** text field, type `10*cos(18*pi/180)[m]`.

9 Right-click **Block 1 (blk1)** and choose **Build Selected**.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **sph1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **blk1** only.

6 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

Convert to Surface 1 (csur1)

1 On the **Geometry** toolbar, click **Conversions** and choose **Convert to Surface**.

2 Select the object **dif1** only.

3 Right-click **Convert to Surface 1 (csur1)** and choose **Build Selected**.

Delete Entities 1 (del1)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **csur1**, select Boundaries 1–8 only.

You can do this by first selecting all boundaries and then removing Boundary 9.

3 Right-click **Component 1 (comp1)>Geometry 1>Delete Entities 1 (del1)** and choose **Build Selected**.

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

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Steel` in the **Label** text field.

3 In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`68.25e6`	Pa	Basic
Poisson's ratio	nu	`0.3`	1	Basic
Density	rho	`6850`	kg/m³	Basic

Note that the density is not used for a static analysis so the value you enter has no effect on the solution.

SHELL (SHELL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.

2 In the **Settings** window for **Shell**, locate the **Thickness** section.

3 In the d text field, type `0.04`.

Symmetry 1

1 Right-click **Component 1 (comp1)>Shell (shell)** and choose **More Constraints>Symmetry**.

2 Select Edges 1 and 4 only.

Prescribed Displacement/Rotation 1

1 In the **Model Builder** window, right-click **Shell (shell)** and choose **Points>Prescribed Displacement/Rotation**.

2 Select Point 4 only.

3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in z direction** check box.

Point Load 1

1 Right-click **Shell (shell)** and choose **Points>Point Load**.

2 Select Point 4 only.

3 In the **Settings** window for **Point Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_\mathrm{P}$ vector as

`-100*para`	x
`0`	y
`0`	z

Point Load 2

1 Right-click **Shell (shell)** and choose **Points>Point Load**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_\mathrm{P}$ vector as

0	x
100*para	y
0	z

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 1 and 4 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 16.

6 In the **Element ratio** text field, type 3.

7 From the **Distribution method** list, choose **Geometric sequence**.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 2 and 3 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 16.

6 In the **Element ratio** text field, type 3.

7 Select the **Symmetric distribution** check box.

8 From the **Distribution method** list, choose **Geometric sequence**.

9 Click **Build All**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
para	0	0	Solver parameter

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Study Settings** section.

3 Select the **Include geometric nonlinearity** check box.

Set up an auxiliary continuation sweep for the **para** parameter.

4 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list
para	range(0,0.1,1)

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Stationary Solver 1**.

4 In the **Settings** window for **Stationary Solver**, locate the **General** section.

5 In the **Relative tolerance** text field, type 0.0001.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node, then click **Fully Coupled 1**.

7 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.

8 Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Constant (Newton)**.

9 In the **Damping factor** text field, type `1`.

10 On the **Study** toolbar, click **Compute**.

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Stress (shell)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Select the **Manual color range** check box.

4 In the **Maximum** text field, type `5e5`.

5 On the **Stress (shell)** toolbar, click **Plot**.

Point Graph 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Point Graph**.

3 Select Point 4 only.

4 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

5 In the **Expression** text field, type `-u`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `para*100[N]`.

8 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `3`.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
`-u under x force`

Point Graph 2

1 Right-click **Results>1D Plot Group 4>Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **Selection** section.

3 Select the **Active** toggle button.

4 In the list, select **4**.

5 Click **Remove from Selection**.

6 Select Point 2 only.

7 Locate the **y-Axis Data** section. In the **Expression** text field, type `v`.

8 Locate the **Legends** section. In the table, enter the following settings:

Legends
`v under y force`

1D Plot Group 4

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** check box.

4 In the associated text field, type `Force (N)`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Displacement under force (m)`.

7 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

8 On the **1D Plot Group 4** toolbar, click **Plot**.

Evaluate the displacements in the points where a comparison should be made with the target.

Point Evaluation 1

1 On the **Results** toolbar, click **Point Evaluation**.

2 Select Points 2 and 4 only.

3 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

4 From the **Parameter selection (para)** list, choose **Last**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`u`	`m`	`Displacement field, X component`

6 Click **Evaluate**.

7 In the table, enter the following settings:

Expression	**Unit**	**Description**
`v`	`m`	`Displacement field, Y component`

8 Click **Evaluate**.

Postbuckling Analysis of a Hinged Cylindrical Shell

Introduction

Buckling is a phenomenon that can cause sudden failure of a structure.

A linear buckling analysis predicts the critical buckling load. Such an analysis, however, does not give any information about what happens at loads higher than the critical load. Tracing the solution after the critical load is called a *postbuckling analysis*.

A linear buckling analysis also often overpredicts the load-carrying capacity of the structure.

In order to accurately determine the critical buckling load or predict the postbuckling behavior, you can use the nonlinear solver and ramp up the applied load to compute the structure deformation. The buckling load can then be based on when a certain, not acceptable, deformation is reached.

Once the critical buckling load has been reached it can happen that the structure undergoes a sudden large deformation into a new stable configuration. This is known as a snap-through phenomenon. A snap-through process cannot be simulated using prescribed load in a standard nonlinear static solver because the problem becomes numerically singular. Physically speaking, it is a highly transient problem as the structure "jumps" from one state to another. For simple cases with a single point load, it is often possible to replace the point load with a prescribed displacement and then measure the reaction force instead.

For more general problems the post-buckling solution must however be tracked using more sophisticated methods, as shown in this example.

Figure 1 shows the variation of load versus the displacement for such a difficult case. It illustrates the possible computational problem by using either a load control (path A) or a displacement control (path B).

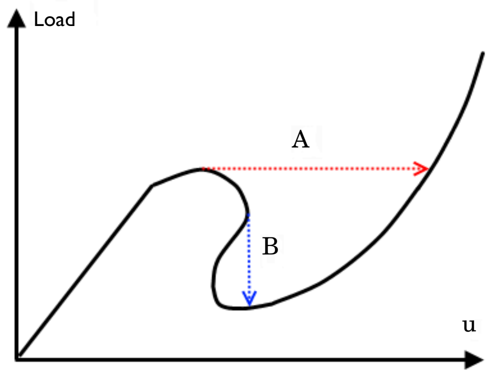


Figure 1: Load versus displacement in snap-through buckling

The shell structure in this example has a behavior similar to this.

Model Definition

The model studied here is a benchmark for a hinged cylindrical panel subjected to a point load at its center; see Ref. 1.

- The radius of the cylinder is $R = 2.54$ m and all edges have a length of $2L = 0.508$ m. The angular span of the panel is thus 0.2 radians. The panel thickness is $th = 6.35$ mm.
- The straight edges are hinged.
- In the study the variation of the panel center vertical displacement with respect to the change of the applied load is of interest.

Due to the double symmetry, only one quarter of the geometry is modeled as shown in Figure 2. The blue lines show the symmetry edge conditions, while the red line shows the location of the hinged edge condition.

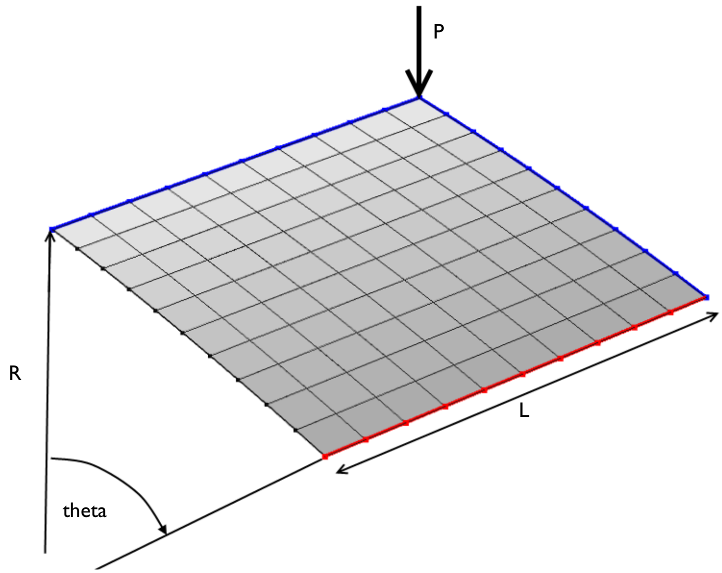


Figure 2: Problem description.

In general, you should be careful with using symmetry in buckling problems, because nonsymmetric solutions may exist.

Results

In Figure 3 you can see the applied load as a function of the panel center displacement. The figure shows clearly a non-unique solution for a given applied load (between -400 N to 600 N) or a given displacement (between 14.4 mm and 17 mm).

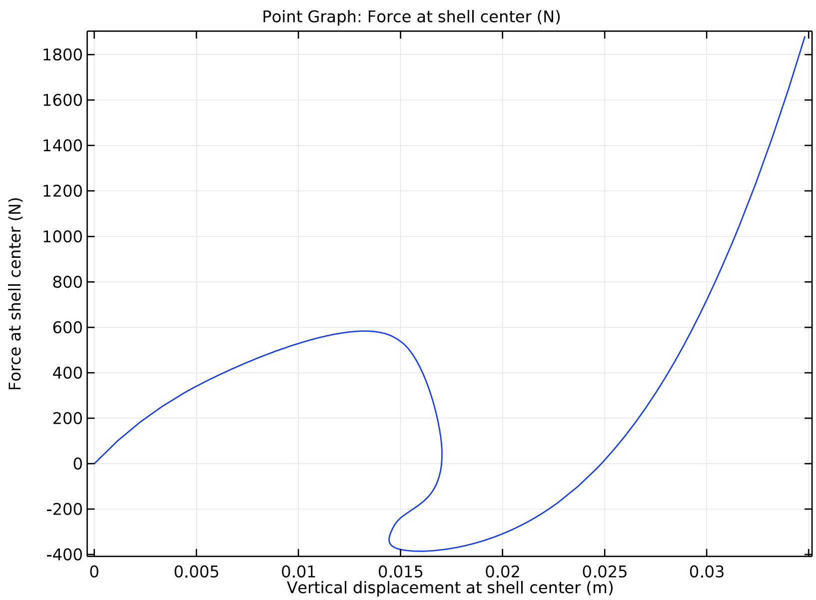


Figure 3: Applied load versus panel center displacement.

As shown in Table 1, the results agree well with the target data from Ref. 1.

TABLE 1: COMPARISON BETWEEN TARGET AND COMPUTED DATA.

Applied Load (N)	Displacement target (mm)	Displacement computed (mm)	Difference (%)
155.1	1.846	1.818	1.52
574.2	11.904	12.05	1.23
485.1	15.501	15.56	0.38
24.9	17.008	17.028	0.12
-300.3	14.520	14.537	0.12
-381.3	16.961	16.77	1.13
-1.8	24.824	24.81	0.06
1469.4	33.388	33.34	0.14

Notes About the COMSOL Implementation

The main feature of this model is that a limit point instability occurs at the buckling load. Neither a load control, nor a point displacement control, would be able to track the jump between the stable solution paths (see Figure 1). To solve this type of problem it is important to find a proper parameter that increases monotonously.

In this example, a good such parameter is the average of the displacement in the direction of the applied force. You use an average coupling operator to measure the displacement and then add a global equation to compute the appropriate point load for each prescribed parameter value.

There is no general way to determine which controlling parameter to use, so it is necessary to use some physical insight.

Reference

1. K.Y. Sze, X.H. Liua, and S.H. Lob, "Popular Benchmark Problems for Geometric Nonlinear Analysis of Shells," *Finite Element in Analysis and Design*, vol. 40, issue 11, pp. 1551–1569, 2004.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/postbuckling_shell`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, click **3D**.
2. In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
R	2540[mm]	2.54 m	Panel radius
L	254[mm]	0.254 m	Panel length
th	6.35[mm]	0.00635 m	Panel thickness
theta	0.1[rad]	0.1 rad	Panel section angle
E0	3.103[GPa]	3.103E9 Pa	Young's modulus
nu0	0.3	0.3	Poisson's ratio
disp	0	0	Displacement parameter

GEOMETRY 1

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

4 Click **Show Work Plane**.

Bézier Polygon 1 (b1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **yw** to R.

5 In row **2**, set **xw** to L and **yw** to R.

6 Right-click **Bézier Polygon 1 (b1)** and choose **Build Selected**.

7 In the **Model Builder** window, click **Geometry 1**.

Revolve 1 (rev1)

1 On the **Geometry** toolbar, click **Revolve**.

2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 In the **End angle** text field, type `theta`.

5 Locate the **Revolution Axis** section. Find the **Direction of revolution axis** subsection. In the **xw** text field, type `1`.

6 In the **yw** text field, type `0`.

7 Right-click **Revolve 1 (rev1)** and choose **Build Selected**.

DEFINITIONS

Click the **Zoom Extents** button on the **Graphics** toolbar.

Average 1 (aveop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 1 only.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 4 only.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	**Expression**	**Unit**
w_center	-intop1(w)	m

SHELL (SHELL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.

2 In the **Settings** window for **Shell**, locate the **Thickness** section.

3 In the d text field, type `th`.

Linear Elastic Material 1

1 In the **Model Builder** window, expand the **Shell (shell)** node, then click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.

3 From the E list, choose **User defined**. In the associated text field, type `E0`.

4 From the ν list, choose **User defined**. In the associated text field, type `nu0`.

Symmetry 1

1 In the **Model Builder** window, right-click **Shell (shell)** and choose **More Constraints>Symmetry**.

2 Select Edge 3 only.

3 In the **Settings** window for **Symmetry**, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Global coordinate system**.

Symmetry 2

1 Right-click **Shell (shell)** and choose **More Constraints>Symmetry**.

2 Select Edge 4 only.

3 In the **Settings** window for **Symmetry**, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Global coordinate system**.

5 Locate the **Symmetry** section. From the **Axis to use as symmetry plane normal** list, choose **1**.

Pinned 1

1 Right-click **Shell (shell)** and choose **More Constraints>Pinned**.

2 Select Edge 2 only.

Point Load 1

1 Right-click **Shell (shell)** and choose **Points>Point Load**.

2 Select Point 4 only.

Apply 1/4th of the total load because of the double symmetry used in this model.

3 In the **Settings** window for **Point Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_\mathrm{P}$ vector as

0	x

0	y
-P/4	z

5 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Global Equations 1

1 Right-click **Shell (shell)** and choose **Global>Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
P	aveop1(-w)-disp	0	0	Force at shell center

4 Locate the **Units** section. Find the **Dependent variable quantity** subsection. From the list, choose **Force load (N)**.

5 Find the **Source term quantity** subsection. From the list, choose **Displacement field (m)**.

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Select Boundary 1 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 1 and 2 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 10.

5 Click **Build Selected**.

STUDY 1

Step 1: Stationary

Set up an auxiliary continuation sweep for the **disp** parameter.

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.

3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list
disp	`range(0,2e-4,1)`

6 Locate the **Study Settings** section. Select the **Include geometric nonlinearity** check box.

Sometimes it is not straightforward to guess the maximum value of the parameter used. You can then instead set a stop condition for the parametric solver based on something that is known.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node.

4 Right-click **Parametric 1** and choose **Stop Condition**.

5 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.

6 Click **Add**.

7 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
`comp1.w_center>0.035`	true	√	`Stop expression 1`

Specify that the solution is to be stored just before the stop condition is reached.

8 Locate the **Output at Stop** section. From the **Add solution** list, choose **Step before stop**.

9 Clear the **Add warning** check box.

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Stationary Solver 1**.

11 In the **Settings** window for **Stationary Solver**, click to expand the **Output** section.

12 Clear the **Reaction forces** check box.

13 Click **Compute**.

RESULTS

Point Graph 1

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Point Graph**.
3. Select Point 4 only.
4. In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Shell>P - Force at shell center**.
5. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6. In the **Expression** text field, type `w_center`.
7. Select the **Description** check box.
8. In the associated text field, type `Vertical displacement at shell center`.
9. On the **1D Plot Group 4** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.3

Scordelis-Lo Roof Shell Benchmark

Introduction

In the following example you build and solve a 3D shell model using the Shell interface. This example is a widely used benchmark model called 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 shown in Figure 1. Only one quarter is analyzed due to symmetry.

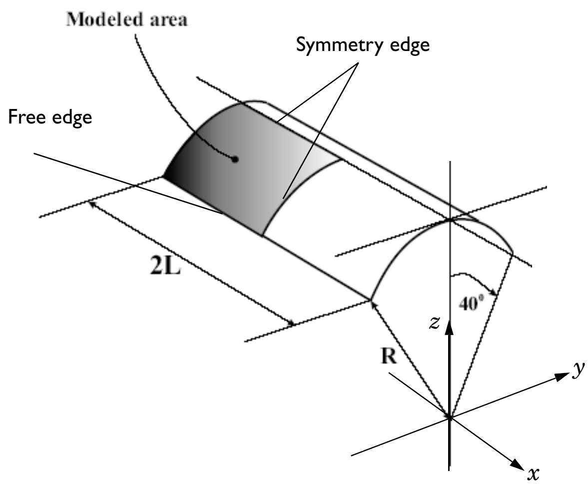


Figure 1: The Scordelis-Lo roof shell benchmark geometry.

- Roof length $2L = 50$ m
- Roof radius $R = 25$ m.

MATERIAL

- Isotropic material with Young's modulus set to $E = 4.32 \cdot 10^8$ N/m^2.
- Poisson's ratio set to $\nu = 0.0$.

CONSTRAINTS

- The outer straight edge is free.
- The outer curved edge is constrained against translation in the y and z directions.
- The straight edge on the top of the roof has symmetry edge constraints.
- The curved inner edge also has symmetry constraints.

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 shown in Figure 2. The computed value is –0.303 m.

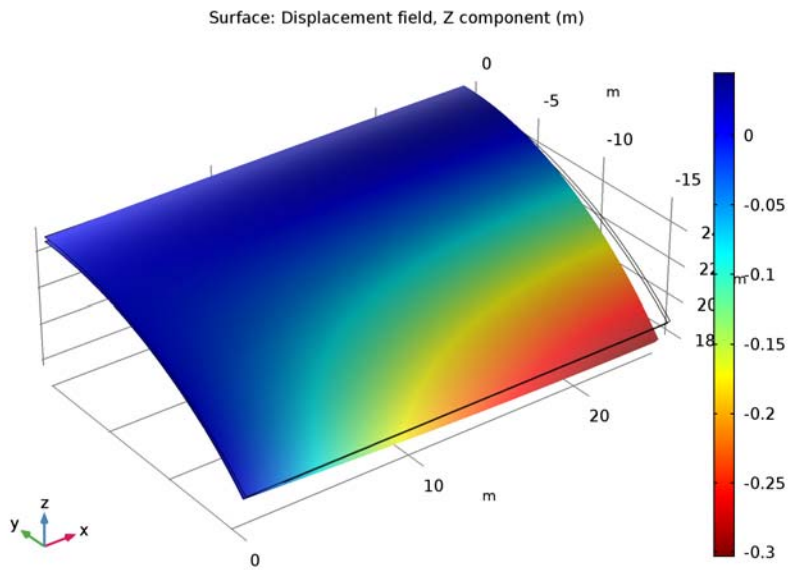


Figure 2: z-displacement with 176 triangular elements.

When changing to a mapped mesh, the more efficient quadrilateral elements are used. The result is –0.301 m as shown in Figure 3. With a very fine mesh, the value converges to –0.302 m, Figure 4. The reference solution quoted in Ref. 1 for the midside vertical displacement is –0.3086 m. The value –0.302 m is in fact observed in other published benchmark results treating this problem as the value that this problem converges towards.

A summary of the performance for different element types and mesh densities is given in Table 1. As can be seen the results are good even with rather coarse meshes.

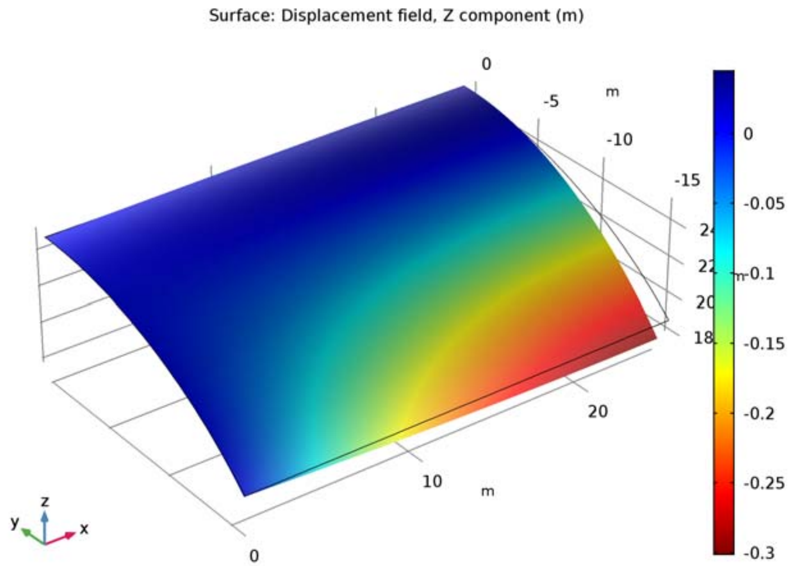


Figure 3: z-displacement with 70 quadrilateral elements.

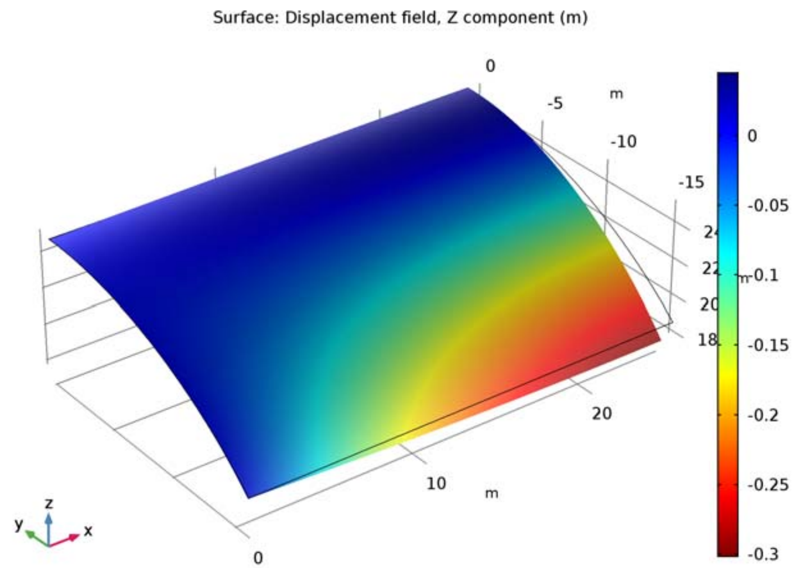


Figure 4: z-displacement with 580 quadrilateral elements.

TABLE 1: CONVERGENCE OF MIDPOINT VERTICAL DISPLACEMENT

MESH SIZE SETTING	ELEMENT TYPE	NUMBER OF ELEMENTS	MIDPOINT DISPLACEMENT
Coarser	Triangle	64	-0.304
Coarser	Quadrilateral	24	-0.300
Normal	Triangle	176	-0.303
Normal	Quadrilateral	70	-0.301
Extra fine	Triangle	1384	-0.302
Extra fine	Quadrilateral	580	-0.302

Reference

1. R.H. MacNeal and R.L. Harder, *Proposed Standard Set of Problems to Test Finite Element Accuracy*, Finite Elements in Analysis and Design, 1, 1985.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/scordelis_lo_roof`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, click **3D**.
2. In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

GEOMETRY 1

Work Plane 1 (wp1)

1. On the **Geometry** toolbar, click **Work Plane**.
2. In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Bézier Polygon 1 (b1)

1. On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.
2. In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.
3. Find the **Added segments** subsection. Click **Add Linear**.
4. Find the **Control points** subsection. In row **1**, set **yw** to 25.
5. In row **2**, set **xw** to 25 and **yw** to 25.
6. Right-click **Bézier Polygon 1 (b1)** and choose **Build Selected**.

Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Revolve 1 (rev1)

1 On the **Geometry** toolbar, click **Revolve**.
2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.
3 Click the **Angles** button.
4 In the **Start angle** text field, type `90`.
5 In the **End angle** text field, type `90+40`.
6 Locate the **Revolution Axis** section. Find the **Direction of revolution axis** subsection. In the **xw** text field, type `1`.
7 In the **yw** text field, type `0`.
8 Right-click **Revolve 1 (rev1)** and choose **Build Selected**.
9 Click the **Zoom Extents** button on the **Graphics** toolbar.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

SHELL (SHELL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.
2 In the **Settings** window for **Shell**, locate the **Thickness** section.
3 In the d text field, type `0.25`.

Symmetry 1

1 Right-click **Component 1 (comp1)>Shell (shell)** and choose **More Constraints>Symmetry**.
2 Select Edges 3 and 4 only.

Prescribed Displacement/Rotation 1

1 In the **Model Builder** window, right-click **Shell (shell)** and choose **Prescribed Displacement/Rotation**.
2 Select Edge 1 only.
3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.
4 Select the **Prescribed in y direction** check box.
5 Select the **Prescribed in z direction** check box.

Face Load 1

1 Right-click **Shell (shell)** and choose **Face and Volume Loads>Face Load**.
2 Click in the **Graphics** window and then press Ctrl+A to select all boundaries.

3 In the **Settings** window for **Face Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	x
0	y
-90	z

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	4.32e8	Pa	Basic
Poisson's ratio	nu	0	1	Basic
Density	rho	1	kg/m³	Basic

MESH 1

First, compute the results with the default triangular mesh.

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Click **Build All**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Stress (shell)

1 In the **Model Builder** window, under **Results** click **Stress (shell)**.

2 In the **Settings** window for **3D Plot Group**, type `Vertical displacement` in the **Label** text field.

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

Surface 1

1 In the **Model Builder** window, expand the **Results>Vertical displacement** node, then click **Surface 1**.

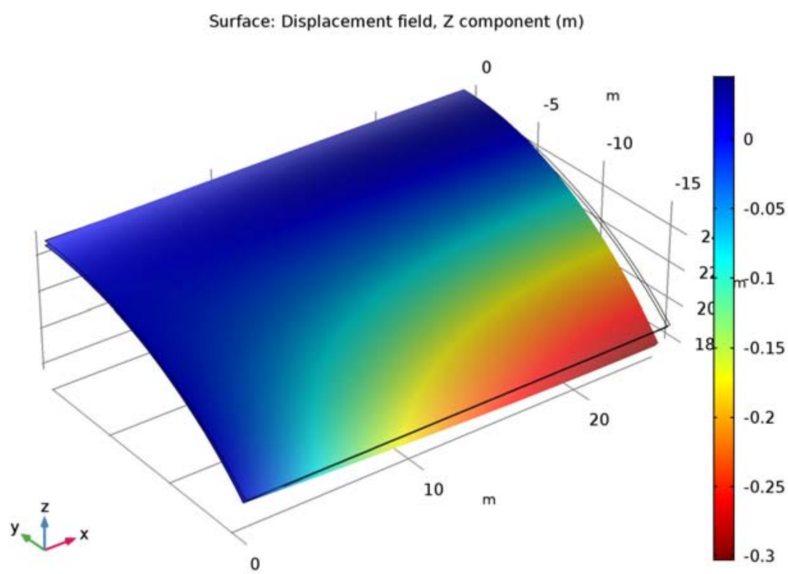
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Shell> Displacement>Displacement field>w - Displacement field, Z component**.

3 Locate the **Coloring and Style** section. Select the **Reverse color table** check box.

Vertical displacement

1 In the **Model Builder** window, under **Results** click **Vertical displacement**.

2 On the **Vertical displacement** toolbar, click **Plot**.



Study 1/Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

2 In the **Settings** window for **Solution**, type `Tri Normal` in the **Label** text field.

Switch to the more effective quadrilateral mesh elements.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, type `Tri Normal` in the **Label** text field.

COMPONENT 1 (COMP1)

Mesh 2

On the **Mesh** toolbar, click **Add Mesh**.

MESH 2

1 In the **Settings** window for **Mesh**, type `Quad Normal` in the **Label** text field.

2 Right-click **Component 1 (comp1)>Meshes>Quad Normal** and choose **More Operations>Mapped**.

QUAD NORMAL

Mapped 1

1 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.

2 From the **Geometric entity level** list, choose **Remaining**.

3 Click **Build All**.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 On the **Home** toolbar, click **Compute**.

RESULTS

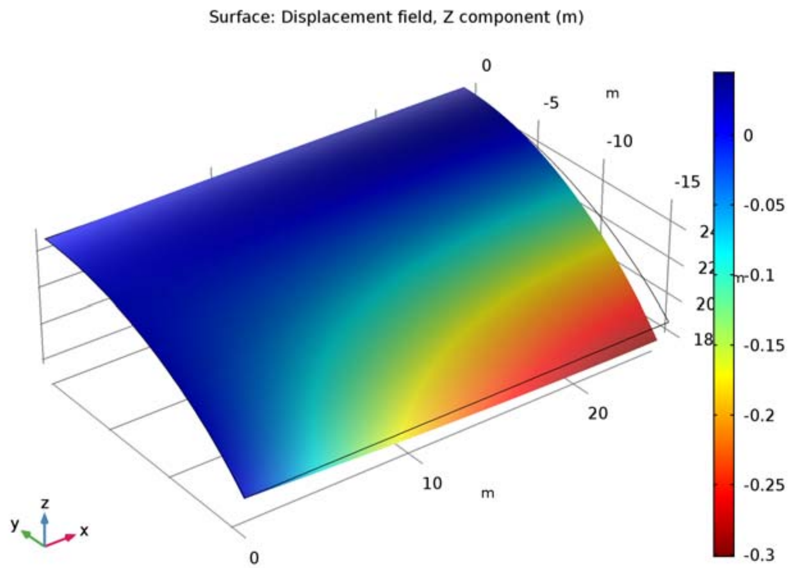
Vertical displacement

1 In the **Model Builder** window, under **Results** click **Vertical displacement**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 On the **Vertical displacement** toolbar, click **Plot**.



Study 2/Solution 2 (sol2)

1 In the **Model Builder** window, under **Results>Data Sets** click **Study 2/Solution 2 (sol2)**.

2 In the **Settings** window for **Solution**, type `Quad Normal` in the **Label** text field.

Examine a well converged result with a fine quadrilateral mesh.

QUAD NORMAL

In the **Model Builder** window, under **Component 1 (comp1)>Meshes** right-click **Quad Normal** and choose **Duplicate**.

QUAD NORMAL 1

In the **Settings** window for **Mesh**, type `Quad Extra fine` in the **Label** text field.

QUAD EXTRA FINE

Size

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Meshes>Quad Extra fine** node, then click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

4 Click **Build All**.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 On the **Home** toolbar, click **Compute**.

RESULTS

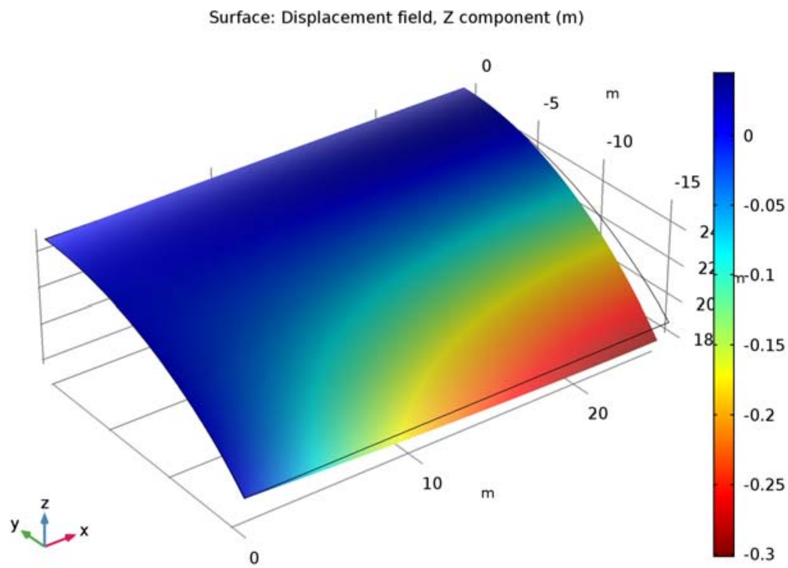
Vertical displacement

1 In the **Model Builder** window, under **Results** click **Vertical displacement**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 3/Solution 3 (sol3)**.

4 On the **Vertical displacement** toolbar, click **Plot**.



Study 3/Solution 3 (sol3)

1 In the **Model Builder** window, under **Results>Data Sets** click **Study 3/Solution 3 (sol3)**.

2 In the **Settings** window for **Solution**, type `Quad Extra fine` in the **Label** text field.

Examine a well converged result with triangles.

TRI NORMAL

In the **Model Builder** window, under **Component 1 (comp1)>Meshes** right-click **Tri Normal** and choose **Duplicate**.

TRI NORMAL 1

In the **Settings** window for **Mesh**, type `Tri Extra Fine` in the **Label** text field.

TRI EXTRA FINE

Size

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Meshes>Tri Extra Fine** node, then click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

4 Click **Build All**.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 4

1 In the **Model Builder** window, click **Study 4**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 On the **Home** toolbar, click **Compute**.

RESULTS

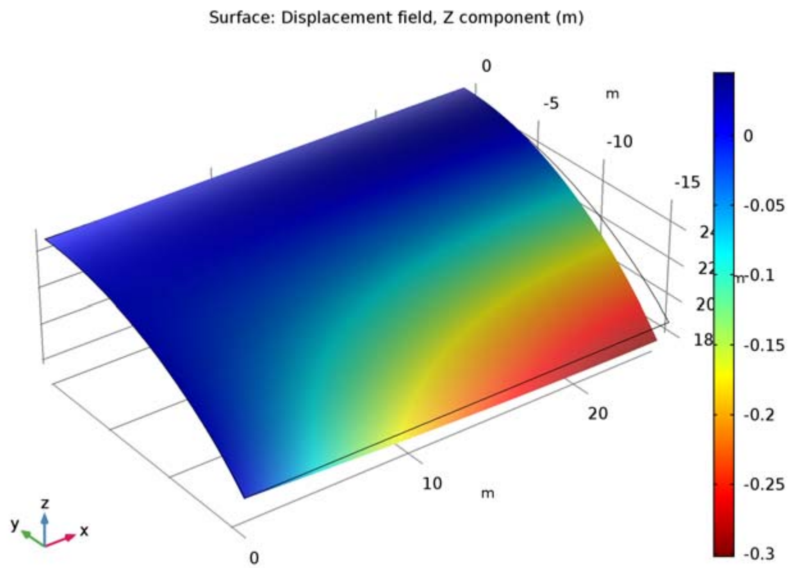
Vertical displacement

1 In the **Model Builder** window, under **Results** click **Vertical displacement**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 4/Solution 4 (sol4)**.

4 On the **Vertical displacement** toolbar, click **Plot**.



Study 4/Solution 4 (sol4)

1 In the **Model Builder** window, under **Results>Data Sets** click **Study 4/Solution 4 (sol4)**.

2 In the **Settings** window for **Solution**, type `Tri Extra fine` in the **Label** text field.

Investigate how well the elements perform with a very coarse mesh.

TRI NORMAL

In the **Model Builder** window, under **Component 1 (comp1)>Meshes** right-click **Tri Normal** and choose **Duplicate**.

TRI NORMAL 1

In the **Settings** window for **Mesh**, type `Tri Coarser` in the **Label** text field.

TRI COARSER

Size

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Meshes>Tri Coarser** node, then click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Coarser**.

4 Click **Build All**.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 5

1 In the **Model Builder** window, click **Study 5**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 On the **Home** toolbar, click **Compute**.

RESULTS

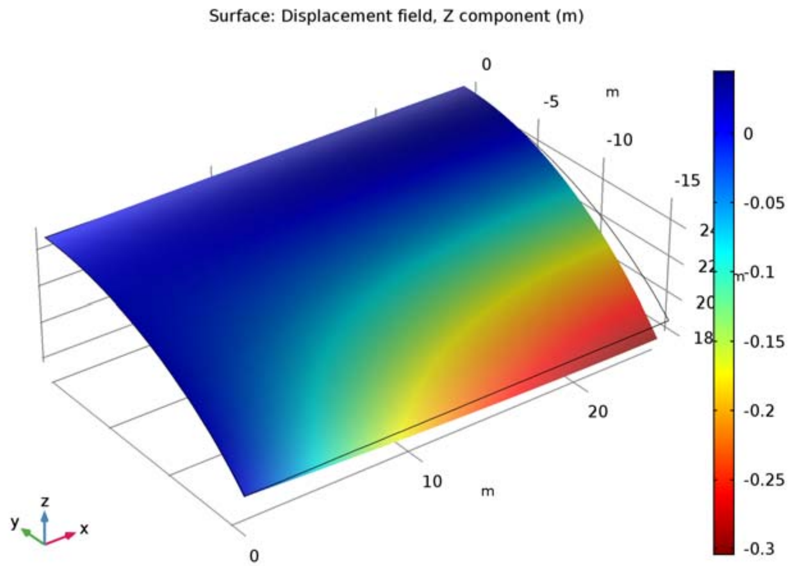
Vertical displacement

1 In the **Model Builder** window, under **Results** click **Vertical displacement**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 5/Solution 5 (sol5)**.

4 On the **Vertical displacement** toolbar, click **Plot**.



Study 5/Solution 5 (sol5)

1 In the **Model Builder** window, under **Results>Data Sets** click **Study 5/Solution 5 (sol5)**.

2 In the **Settings** window for **Solution**, type `Tri Coarser` in the **Label** text field.

QUAD NORMAL

In the **Model Builder** window, under **Component 1 (comp1)>Meshes** right-click **Quad Normal** and choose **Duplicate**.

QUAD NORMAL 1

In the **Settings** window for **Mesh**, type `Quad Coarser` in the **Label** text field.

QUAD COARSER

Size

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Meshes>Quad Coarser** node, then click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Coarser**.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 6

1 In the **Model Builder** window, click **Study 6**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 On the **Home** toolbar, click **Compute**.

RESULTS

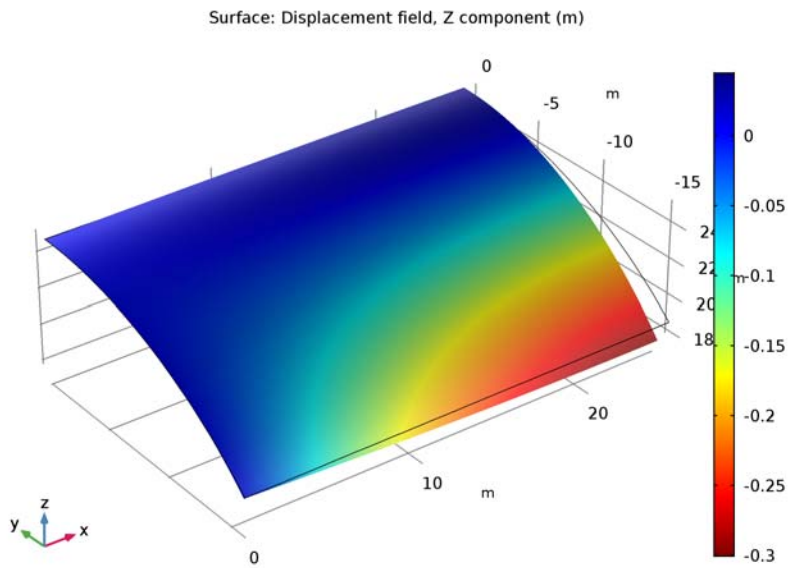
Vertical displacement

1 In the **Model Builder** window, under **Results** click **Vertical displacement**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 6/Solution 6 (sol6)**.

4 On the **Vertical displacement** toolbar, click **Plot**.



Study 6/Solution 6 (sol6)

1 In the **Model Builder** window, under **Results>Data Sets** click **Study 6/Solution 6 (sol6)**.

2 In the **Settings** window for **Solution**, type `Quad Coarser` in the **Label** text field.

Created in COMSOL Multiphysics 5.3

Single Edge Crack

Introduction

This example deals with the stability of a plate with an edge crack that is subjected to a tensile load. To analyze the stability of existing cracks, you can apply the principles of 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 causes the plate to fracture. When this calculated value becomes equal to the critical fracture toughness of the material, K_{Ic} (a material property), then usually catastrophic fracture occurs.

Model Definition

A plate with a width of 1.5 m and height of 3 m has a single horizontal edge-crack of length a = 0.6 m on the left vertical edge, see Figure 1. En external load is pulling the plate such that top and bottom edges experience tensile stress, σ, of 20 MPa.

Due to symmetry reasons only half of the plate is modeled. Additional domains are created in the half plate rectangle to create path for integration contours of the stress intensity factor.

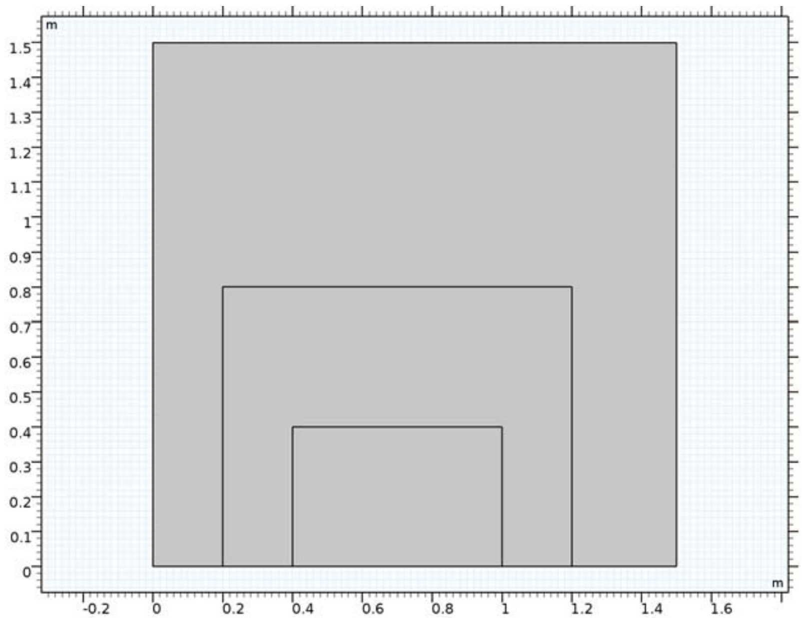


Figure 1: Plate geometry.

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.

MATERIAL MODEL

Due to the interior boundaries the geometry consists of three domains. The same material properties apply to all three domains:

TABLE 0-1:

QUANTITY	NAME	EXPRESSION
Young's modulus	E	$206 \cdot 10^9$
Poisson's ratio	ν	0.3

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:

$$J = \frac{K_I^2}{E} \tag{1}$$

where E is Young's modulus.

Results

Based on Ref. 1 an analytical solution for the stress intensity factor is

$$K_{\mathrm{Ia}} = \sigma \cdot \sqrt{\pi \cdot a} \cdot \mathrm{ccf}$$

where $\sigma = 20$ MPa (edge stress), $a = 0.6$ m (crack length), and $\mathrm{ccf} = 2.1$ (configuration correction factor). This correction factor is calculated with an polynomial equation from Ref. 1. The above values gives the stress intensity factor $K_{\mathrm{Ia}} = 57.66\ \mathrm{MPa{\cdot}m}^{1/2}$.

The calculated stress intensity factors for the three different contours is

CONTOUR	STRESS INTENSITY FACTOR
1	57.63 $\mathrm{MPa{\cdot}m}^{1/2}$
2	57.63 $\mathrm{MPa{\cdot}m}^{1/2}$
3	57.62 $\mathrm{MPa{\cdot}m}^{1/2}$

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.

Figure 2 shows the stress singularity at the crack tip.

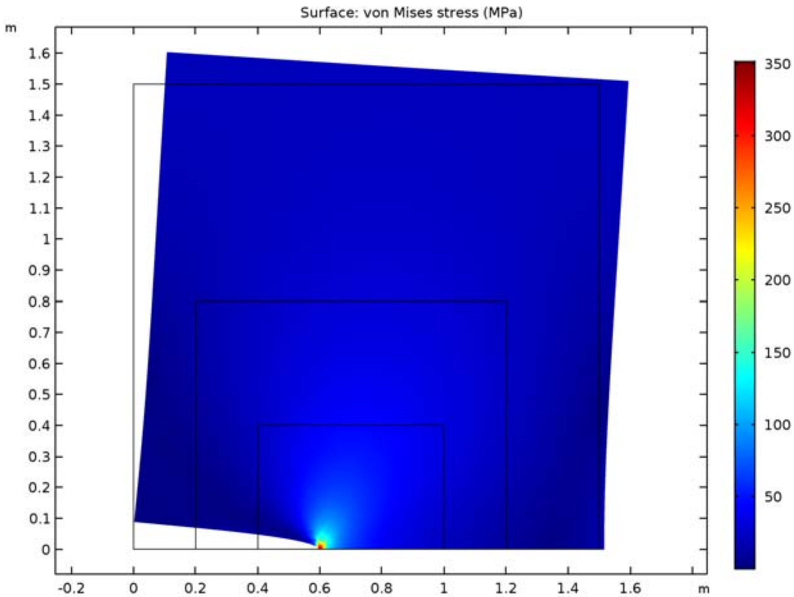


Figure 2: von Mises stresses and the deformed shape of the plate. The displacement is exaggerated to illustrate the deformation under the applied load.

Notes about the COMSOL Implementation

In this analysis you compute the J-integral 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 integration operators for each contour. You then use these operators when setting up global expressions for the calculation of the stress intensity factors for the contours. The first expression, 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. Finally, you can compute the stress intensity factor from the J-integral value, according to Equation 1.

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. To make sure that this is the case for all of the involved boundaries, you can define the normals as boundary expressions.

Reference

1. A-R. Ragab and S.E. Bayoumi, *Engineering Solid Mechanics*, CRC Press, 1998.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/single_edge_crack`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`E0`	`2.06e11[Pa]`	2.06E11 Pa	`Young's modulus`

The reason for defining a parameter for Young's modulus is that it appears in some variables that you will define later.

GEOMETRY 1

Square 1 (sq1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Square**.
2. In the **Settings** window for **Square**, locate the **Size** section.
3. In the **Side length** text field, type `1.5`.

Rectangle 1 (r1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Height** text field, type `0.8`.
4. Locate the **Position** section. In the **x** text field, type `0.2`.

Rectangle 2 (r2)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `0.6`.
4. In the **Height** text field, type `0.4`.
5. Locate the **Position** section. In the **x** text field, type `0.4`.

Point 1 (pt1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Point**.
2. In the **Settings** window for **Point**, locate the **Point** section.
3. In the **x** text field, type `0.6`.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Steel` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`E0`	Pa	Basic
Poisson's ratio	nu	`0.3`	1	Basic
Density	rho	`7850`	kg/m³	Basic

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.

3 From the list, choose **Plane stress**.

4 Locate the **Thickness** section. In the d text field, type `0.01`.

Symmetry 1

1 Right-click **Component 1 (comp1)>Solid Mechanics (solid)** and choose **More Constraints>Symmetry**.

2 Select Boundaries 10, 12, and 14 only.

Boundary Load 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Boundary Load**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

`0`	x
`20[MPa]`	y

Prescribed Displacement 1

1 Right-click **Solid Mechanics (solid)** and choose **Points>Prescribed Displacement**.

2 Select Point 7 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

DEFINITIONS

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 1, 3, and 15 only.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 4, 6, and 13 only.

Integration 3 (intop3)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 7, 9, and 11 only.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 1, 4, and 7 only.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression
Nx	-1
Ny	0

Variables 2

1 On the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 3, 6, and 9 only.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression
Nx	0
Ny	1

Variables 3

1 On the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 11, 13, and 15 only.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression
Nx	1
Ny	0

Variables 4

1 On the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
W_1	intop1(solid.Ws*Nx)	J/m^2	Strain energy density, contour 1
Tdudx_1	intop1(-((solid.sx*Nx+ solid.sxy*Ny)*uX+ (solid.sxy*Nx+ solid.sy*Ny)*vX))	N/m	Traction vector times displacement derivative, contour 1
J_1	2*(W_1+Tdudx_1)	N/m	J-integral, contour 1
KI_1	sqrt(E0*abs(J_1))		Stress intensity factor, contour 1
W_2	intop2(solid.Ws*Nx)	J/m^2	Strain energy density, contour 2

Name	Expression	Unit	Description
Tdudx_2	intop2(-((solid.sx*Nx+solid.sxy*Ny)*uX+(solid.sxy*Nx+solid.sy*Ny)*vX))	N/m	Traction vector times displacement derivative, contour 2
J_2	2*(W_2+Tdudx_2)	N/m	J-integral, contour 2
KI_2	sqrt(E0*abs(J_2))		Stress intensity factor, contour 2
W_3	intop3(solid.Ws*Nx)	J/m^2	Strain energy density, contour 3
Tdudx_3	intop3(-((solid.sx*Nx+solid.sxy*Ny)*uX+(solid.sxy*Nx+solid.sy*Ny)*vX))	N/m	Traction vector times displacement derivative, contour 3
J_3	2*(W_3+Tdudx_3)	N/m	J-integral, contour 3
KI_3	sqrt(E0*abs(J_3))		Stress intensity factor, contour 3

MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.
3. From the **Element size** list, choose **Extra fine**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Surface 1

1. In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. From the **Unit** list, choose **MPa**.
4. On the **Stress (solid)** toolbar, click **Plot**.
5. Click the **Zoom Extents** button on the **Graphics** toolbar.

Global Evaluation 1

1. On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Definitions> Variables>KI_1 - Stress intensity factor, contour 1**.

3 Click **Evaluate**.

Global Evaluation 2

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Definitions> Variables>KI_2 - Stress intensity factor, contour 2**.

3 Click **Evaluate**.

Global Evaluation 3

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Definitions> Variables>KI_3 - Stress intensity factor, contour 3**.

3 Click **Evaluate**.

Created in COMSOL Multiphysics 5.3

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 example 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 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.

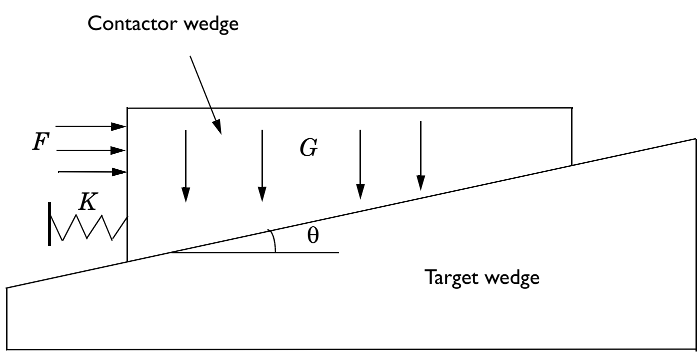


Figure 1: Sliding wedge with linear springs and a boundary load.

The aim of this benchmark is to calculate the horizontal sliding distance and compare it with an elementary statics calculation. Several cases using different friction coefficients are computed, $\mu = 0$; 0.1 0.2.

For each friction coefficient a specific overall spring stiffness, K, is used (K = 1194 N/m; 882 N/m and 563.9 N/m respectively).

The horizontal applied force is F = 1500 N, the total vertical gravity load is G = 3058 N, the wedge angle is $\tan\theta = 0.1$.

For all study cases the expected horizontal sliding distance is expected to be 1m.

The mesh is shown in Figure 2.

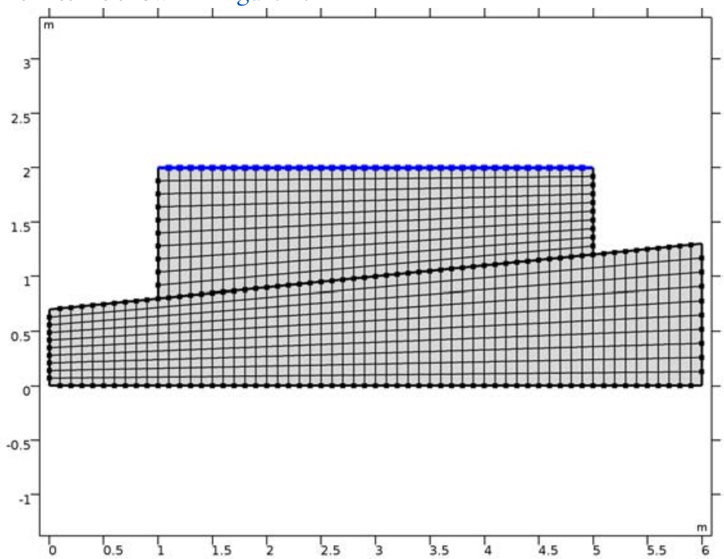


Figure 2: Quadrilateral elements is used to mesh the model.

The total number of elements in this model is 1000 and the number of degrees of freedom is 8967.

Results and Discussion

The horizontal displacement computed for all friction case agree very well with the reference data, see Ref. 1. For all case the difference is lower than 0.1%.

Figure 3 below shows the result for the case μ = *0.2*, *K* = 563.9 N/m.

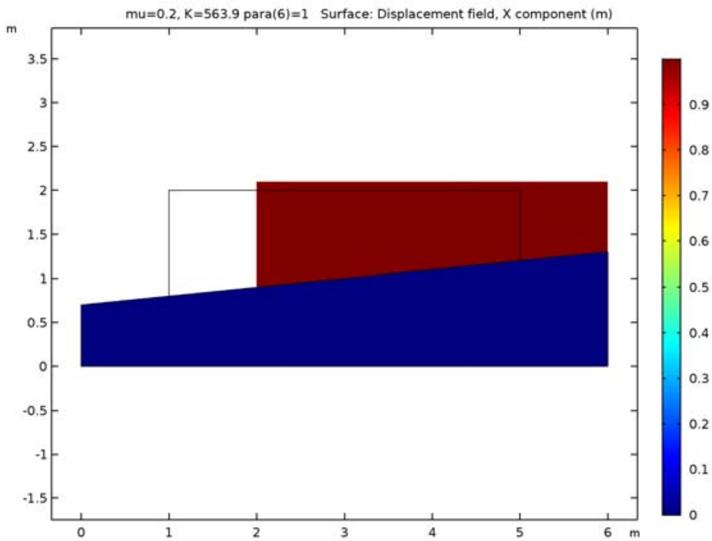


Figure 3: A surface plot of the x-displacement of the contactor wedge.

Reference

1. Feng Q., *NAFEMS Benchmark Tests for Finite Element Modelling of Contact, Gapping and Sliding*. NAFEMS Ref. R0081, UK, 2001.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/sliding_wedge`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
G	3058[N]	3058 N	Gravity load
F	1500[N]	1500 N	Applied force
K	0[N/m]	0 N/m	Spring stiffness
mu	0	0	Friction coefficient
L	1.2[m]	1.2 m	Edge length
para	0	0	Computation parameter

GEOMETRY 1

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **x** to `6`.

5 Find the **Added segments** subsection. Click **Add Linear**.

6 Find the **Control points** subsection. In row **2**, set **y** to `1.3`.

7 Find the **Added segments** subsection. Click **Add Linear**.

8 Find the **Control points** subsection. In row **2**, set **x** to `0` and **y** to `0.7`.

9 Find the **Added segments** subsection. Click **Add Linear**.

10 Find the **Control points** subsection. Click **Close Curve**.

11 Click **Build All Objects**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `4`.
4 In the **Height** text field, type `1.2`.
5 Locate the **Position** section. In the **x** text field, type `1`.
6 In the **y** text field, type `0.8`.

Copy 1 (copy1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Copy**.
2 Select the object **b1** only.
3 Right-click **Copy 1 (copy1)** and choose **Build Selected**.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2 Select the object **r1** only.
3 In the **Settings** window for **Difference**, locate the **Difference** section.
4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
5 Select the object **copy1** only.
6 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.
2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.
3 From the **Action** list, choose **Form an assembly**.
4 From the **Pair type** list, choose **Contact pair**.
5 Right-click **Component 1 (comp1)>Geometry 1>Form Union (fin)** and choose **Build Selected**.
6 Click the **Zoom Extents** button on the **Graphics** toolbar.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`206[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.3`	1	Basic
Density	rho	`6000[kg/m^3]`	kg/m³	Basic

SOLID MECHANICS (SOLID)

Body Load 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Volume Forces>Body Load**.

2 Select Domain 2 only.

3 In the **Settings** window for **Body Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{tot}$ vector as

`0`	x
`-G*para`	y

Contact 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Pairs>Contact**.

2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.

3 In the **Pairs** list, select **Contact Pair 1 (ap1)**.

Since the two pieces can be expected to be in contact always, the convergence rate cane be increased by using a less conservative value of the penalty factor.

4 Locate the **Penalty Factor** section. From the **Tuned for** list, choose **Speed**.

Friction 1

1 Right-click **Component 1 (comp1)>Solid Mechanics (solid)>Contact 1** and choose **Friction**.

2 In the **Settings** window for **Friction**, locate the **Friction** section.

3 In the μ_{stat} text field, type `mu`.

4 Locate the **Initial Values** section. From the **Previous contact state** list, choose **In contact**.

Spring Foundation 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose the boundary condition **Mass, Spring, and Damper>Spring Foundation**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

4 From the **Spring type** list, choose **Spring constant per unit length**.

5 From the list, choose **Diagonal**.

6 In the $\mathbf{k}_L$ table, enter the following settings:

K/L	0
0	0

Boundary Load 1

1 Right-click **Solid Mechanics (solid)** and choose **Boundary Load**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{tot}$ vector as

F*para	x
0	y

Fixed Constraint 1

1 Right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 Select Boundary 2 only.

MESH 1

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 Select Boundaries 1 and 5 only.

4 In the **Settings** window for **Distribution**, locate the **Distribution** section.

5 In the **Number of elements** text field, type 10.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 60.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 40.

5 Click **Build All**.

STUDY 1

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
mu	0 0.1 0.2	

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
K	1194 882 563.9	

Step 1: Stationary

Set up an auxiliary continuation sweep for the `para` parameter.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.

3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para	0 0.03 0.2 0.4 0.8 1	

In this example the contact forces are very small, so it is necessary so set proper scales for these variables.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 1** node, then click **Friction force (spatial frame) (comp1.solid.Tt_ap1)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `100`.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Dependent Variables 1** click **Contact pressure (comp1.solid.Tn_ap1)**.

7 In the **Settings** window for **Field**, locate the **Scaling** section.

8 In the **Scale** text field, type `1000`.

9 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1>Segregated 1** node, then click **Segregated Step 1**.

10 In the **Settings** window for **Segregated Step**, click to expand the **Method and termination** section.

11 Locate the **Method and Termination** section. In the **Tolerance factor** text field, type `10`.

12 On the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Displacement (solid)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Displacement (solid)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Displacement>Displacement field (material and geometry frames)>u - Displacement field, X component**.
3 On the **Displacement (solid)** toolbar, click **Plot**.
4 Click the **Zoom Extents** button on the **Graphics** toolbar.

Follow the instructions below to evaluate the horizontal displacement for all three friction case.

Point Evaluation 1

1 On the **Results** toolbar, click **Point Evaluation**.
2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.
3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.
4 From the **Parameter selection (para)** list, choose **Last**.
5 From the **Table columns** list, choose **mu, K**.
6 Select Point 8 only.
7 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Solid Mechanics>Displacement>Displacement field (material and geometry frames)>u - Displacement field, X component**.
8 Click **Evaluate**.

Created in COMSOL Multiphysics 5.3

Thermally Loaded Beam

Introduction

In the following tutorial, you build and solve a 3D beam model using the 3D Beam interface. This example shows how to model a thermally induced deformation of a beam. Temperature gradients are applied between 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 E = 210 GPa.
- Poisson's ratio $\nu = 0.3$.
- Coefficient of thermal expansion $\alpha = 11{\cdot}10^{-6}$/°C.

CONSTRAINTS

- On one end the beam has constrained displacements in all directions and it has the rotation around its length constraint as well to prevent the singular rotational degrees of freedom.
- On the other end the movement perpendicular to the beams length is prescribed.

THERMAL LOAD

Figure 1 shows the surface temperature at each corner of the cross section. 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.

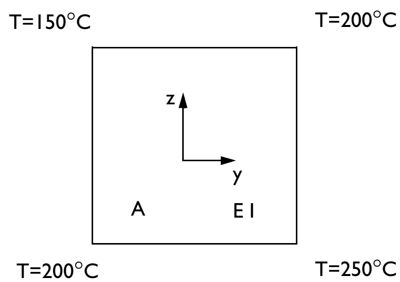


Figure 1: Geometric properties and thermal loads at corners.

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:

$$w = \frac{\alpha L^2}{8t}(T_2 - T_1)$$

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 2 shows the global z-displacement along the beam.

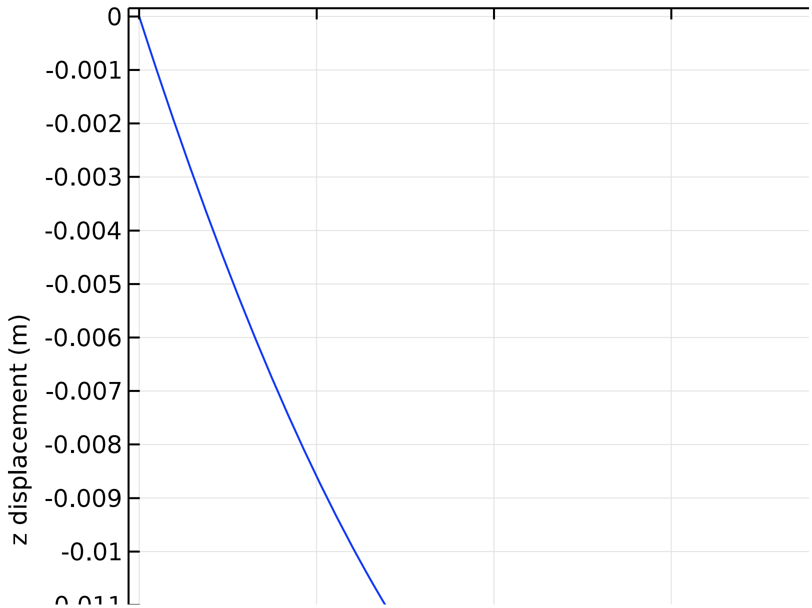


Figure 2: z-displacement along the beam.

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 maximum camber calculated with COMSOL Multiphysics and the analytical value is shown in the table below.

COMSOL Multiphysics	**Analytical**
22.1 mm	21.9 mm

Figure 3 shows the total camber along the beam.

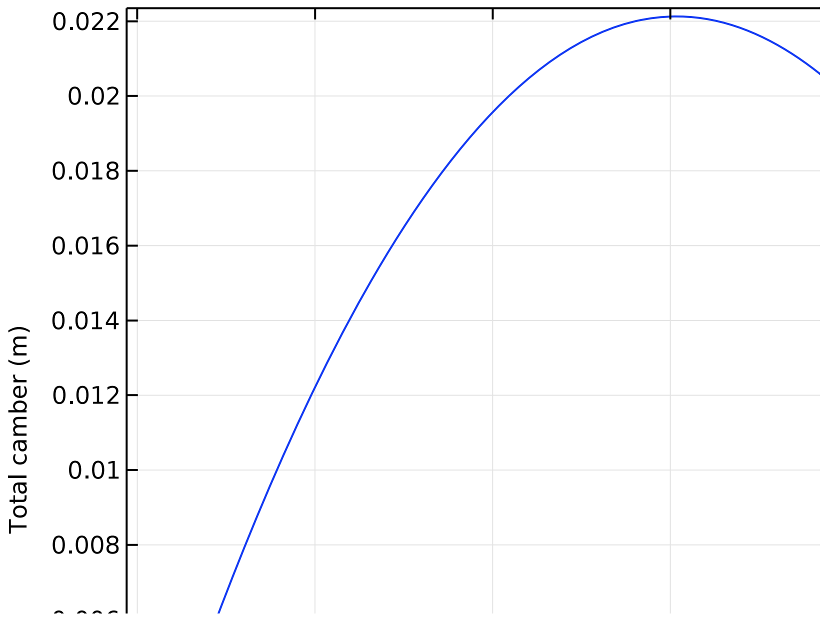


Figure 3: Camber along the beam.

Reference

1. W. Young, *Roark's Formulas for Stress & Strain*, McGraw-Hill, 1989.

Application Library path: `Structural_Mechanics_Module/ Verification_Examples/thermally_loaded_beam`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Beam (beam)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
a	0.04[m]	0.04 m	Side length

GEOMETRY 1

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **x** to 3.

5 Click **Build All Objects**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, click to expand the **Material properties** section.

3 Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Coefficient of Thermal Expansion**.

4 Click **Add to Material**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Coefficient of thermal expansion	alpha	11e-6	1/K	Basic
Young's modulus	E	210e9	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	7800	kg/m³	Basic

BEAM (BEAM)

Cross Section Data 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Beam (beam)** node, then click **Cross Section Data 1**.

2 In the **Settings** window for **Cross Section Data**, locate the **Cross Section Definition** section.

3 From the list, choose **Common sections**.

4 In the h_y text field, type a.

5 In the h_z text field, type a.

Section Orientation 1

1 In the **Model Builder** window, expand the **Cross Section Data 1** node, then click **Section Orientation 1**.

2 In the **Settings** window for **Section Orientation**, locate the **Section Orientation** section.

3 From the **Orientation method** list, choose **Orientation vector**.

4 Specify the V vector as

0	x
1	y
0	z

Prescribed Displacement/Rotation 1

1 On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.

2 Select Point 1 only.

3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in y direction** check box.

6 Select the **Prescribed in z direction** check box.

7 Locate the **Prescribed Rotation** section. Select the **Prescribed in x direction** check box.

Prescribed Displacement/Rotation 2

1 On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.

2 Select Point 2 only.

3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in y direction** check box.

5 Select the **Prescribed in z direction** check box.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)** click **Linear Elastic Material 1**.

Thermal Expansion 1

1 On the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.

2 In the **Settings** window for **Thermal Expansion**, locate the **Thermal Expansion Properties** section.

3 In the T_{ref} text field, type `0`.

4 Locate the **Thermal Bending** section. In the T_{gy} text field, type `50/0.04`.

5 In the T_{gz} text field, type `-50/0.04`.

6 Locate the **Model Input** section. In the T text field, type `200`.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Stress (beam)

1 In the **Model Builder** window, under **Results** click **Stress (beam)**.

2 In the **Settings** window for **3D Plot Group**, type `Displacements` in the **Label** text field.

Line 1

1 In the **Model Builder** window, expand the **Results>Displacements** node, then click **Line 1**.

2 In the **Settings** window for **Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Beam>Displacement> beam.disp - Total displacement**.

3 On the **Displacements** toolbar, click **Plot**.

Line Graph 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Model Builder** window, right-click **1D Plot Group 9** and choose **Line Graph**.

3 In the **Settings** window for **Line Graph**, locate the **Selection** section.

4 From the **Selection** list, choose **All edges**.

5 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Beam>Displacement>Displacement field>w - Displacement field, z component**.

6 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Component 1>Geometry>Coordinate>x - x-coordinate**.

1D Plot Group 9

1 In the **Model Builder** window, under **Results** click **1D Plot Group 9**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. Select the **y-axis label** check box.

5 In the associated text field, type `z displacement (m)`.

6 On the **1D Plot Group 9** toolbar, click **Plot**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

1D Plot Group 10

1 Right-click **Results>1D Plot Group 9** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 In the **y-axis label** text field, type `Total camber (m)`.

Line Graph 1

1 In the **Model Builder** window, expand the **1D Plot Group 10** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Beam>Displacement> beam.disp - Total displacement**.

3 On the **1D Plot Group 10** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.3

Thick Plate Stress Analysis

Introduction

This example implements the static stress analysis described in the NAFEMS Test No LE10, "Thick Plate Pressure," found on page 77 in the NAFEMS report *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 in geometry, the analysis only includes a quarter of the ellipse.

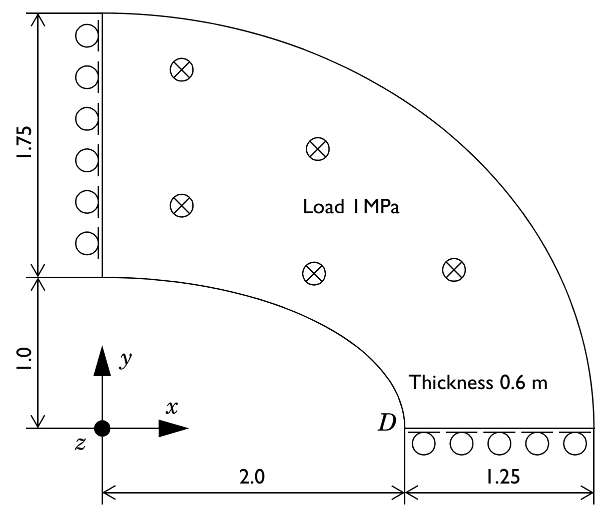


Figure 1: The thick plate geometry, reduced to a quarter of the ellipse due to symmetry.

MATERIAL

Isotropic with $E = 2.1 \cdot 10^{11}$ Pa, $\nu = 0.3$.

LOAD

A distributed load of 10^6 Pa on the upper surface pointing in the negative z direction.

CONSTRAINTS

- Symmetry planes, $x = 0$, $y = 0$.

- Outer ellipse surface constrained in the x and y directions.
- Midplane on outer ellipse surface constrained in the z direction.

Results

The normal stress σ_y is evaluated on the top surface at the inside of the elliptic hole, point D in Figure 1 with coordinate (2, 0, 0.6). It is in good agreement with the NAFEMS benchmark (Ref. 1), considering the coarse mesh. The difference is less than 4%.

RESULT	COMSOL MULTIPHYSICS	NAFEMS (Ref. 1)
σ_y (at D)	-5.57 MPa	-5.38 MPa

The y-component of the stress is shown in Figure 2.

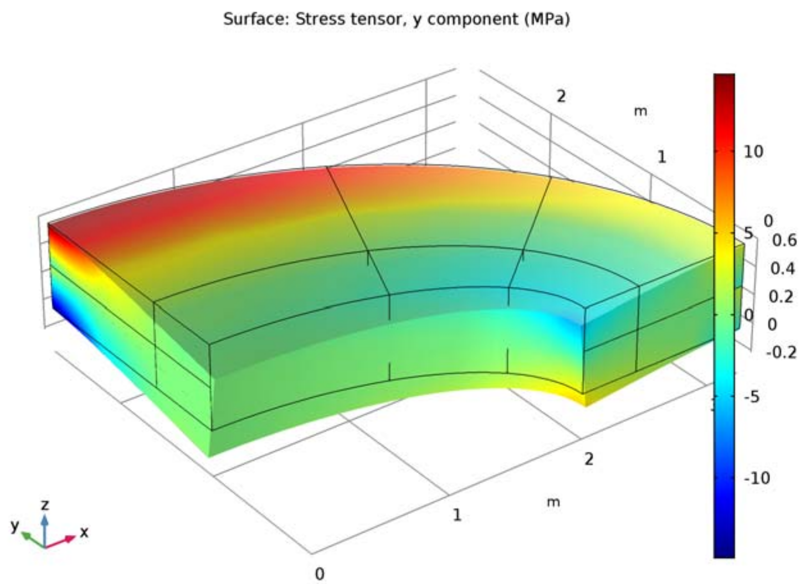


Figure 2: The stress in the y direction.

A note about this example is that the z direction constraint is applied to an edge only. This is a singular constraint, which causes local stresses at the constrained edge. These stresses are unlimited from a theoretical point of view, and in practice the stresses and vertical displacements are strongly mesh dependent. This does not invalidate the possibility to determine stresses at a distance far away from the singular constraint.

Notes About the COMSOL Implementation

In order get the same mesh as in the original benchmark, some extra lines are drawn in the 2D geometry. As an effect, there will be several domains. This approach is efficient in this simple example, whereas for more complex geometries, the use of **Mesh Control Domains** should be considered.

Reference

1. G.A.O. Davies, R.T. Fenner, and R.W. Lewis, *Background to Benchmarks*, NAFEMS, Glasgow, 1993.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/thick_plate`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Ellipse 1 (e1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Ellipse**.

2 In the **Settings** window for **Ellipse**, locate the **Size and Shape** section.

3 In the **a-semiaxis** text field, type `3.25`.

4 In the **b-semiaxis** text field, type `2.75`.

5 In the **Sector angle** text field, type `90`.

6 Right-click **Ellipse 1 (e1)** and choose **Build Selected**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Ellipse 2 (e2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Ellipse**.

2 In the **Settings** window for **Ellipse**, locate the **Size and Shape** section.

3 In the **a-semiaxis** text field, type `2`.

4 In the **Sector angle** text field, type `90`.

5 Right-click **Ellipse 2 (e2)** and choose **Build Selected**.

Ellipse 3 (e3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Ellipse**.

2 In the **Settings** window for **Ellipse**, locate the **Size and Shape** section.

3 In the **a-semiaxis** text field, type `2.416`.

4 In the **b-semiaxis** text field, type `1.583`.

5 In the **Sector angle** text field, type `90`.

6 Right-click **Ellipse 3 (e3)** and choose **Build Selected**.

Difference 1 (dif1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **e1** and **e3** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **e2** only.

6 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

Polygon 1 (pol1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. In the **xw** text field, type `1.783 1.165`.

5 In the **yw** text field, type `2.3 0.812`.

Polygon 2 (pol2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. In the **xw** text field, type `2.833 1.783`.

5 In the **yw** text field, type `1.348 0.453`.

6 On the **Work Plane** toolbar, click **Build All**.

Plane Geometry

Click the **Zoom Extents** button on the **Graphics** toolbar.

Partition Objects 1 (par1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

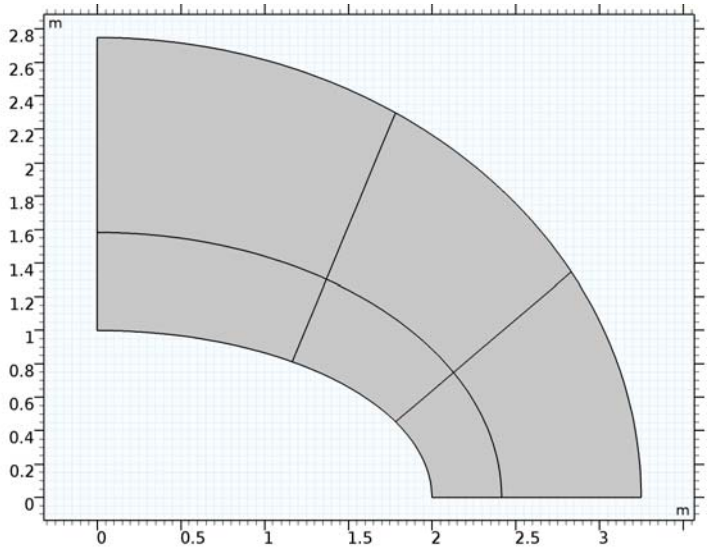
3 Find the **Objects to partition** subsection. Select the **Active** toggle button.

4 Select the object **dif1** only.

5 Find the **Tool objects** subsection. Select the **Active** toggle button.

6 Select the objects **pol1** and **pol2** only.

7 Click **Build Selected**.



Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

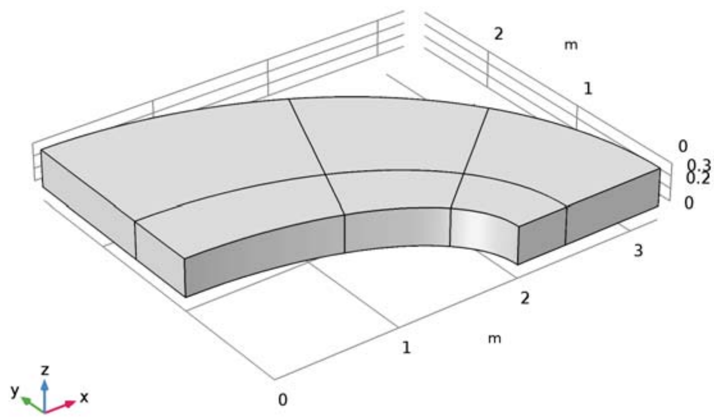
2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
0.3

4 Right-click **Extrude 1 (ext1)** and choose **Build Selected**.

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



Array 1 (arr1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **ext1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **z size** text field, type `2`.

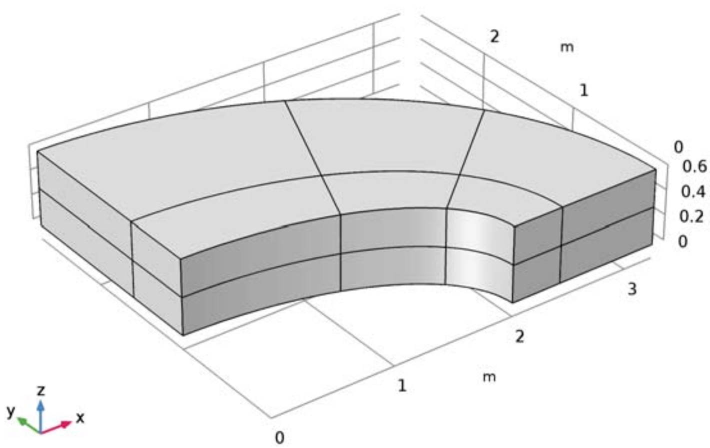
5 Locate the **Displacement** section. In the **z** text field, type `0.3`.

6 Right-click **Array 1 (arr1)** and choose **Build Selected**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.



MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	210[GPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	7850	kg/m³	Basic

SOLID MECHANICS (SOLID)

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1, 4, 8, 11, 40, 41, 49, and 50 only.

Prescribed Displacement 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundaries 15, 16, 31, 32, 51, and 52 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in y direction** check box.

Prescribed Displacement 2

1 On the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement**.

2 Select Edges 20, 41, and 72 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in z direction** check box.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundaries 7, 14, 23, 30, 39, and 48 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	x
0	y
-1e6	z

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 Select Boundaries 7, 14, 23, 30, 39, and 48 only.

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Mapped 1** click **Distribution 1**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 In the **Number of elements** text field, type 2.

4 Locate the **Edge Selection** section. From the **Selection** list, choose **All edges**.

5 Click **Build Selected**.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 In the **Settings** window for **Swept**, click **Build All**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Point Evaluation 1

1 On the **Results** toolbar, click **Point Evaluation**.

2 Select Point 24 only.

This corresponds to point D in Figure 1.

3 In the **Settings** window for **Point Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1> Solid Mechanics>Stress>Stress tensor (spatial frame)>solid.sy - Stress tensor, y component**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
solid.sy	MPa	Stress tensor, y component

5 Click **Evaluate**.

Stress (solid)

Modify the default surface plot to show the y component of the stress tensor.

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress (solid)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Stress> Stress tensor (spatial frame)>solid.sy - Stress tensor, y component**.

3 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

4 On the **Stress (solid)** toolbar, click **Plot**.

Vibrating Membrane

Introduction

In the following example you compute the natural frequencies of a pre-tensioned membrane using the 3D Membrane interface. This is an example of "stress stiffening"; where the transverse stiffness of a membrane is directly proportional to the tensile force.

The results are compared with the analytical solution.

Model Definition

The model consists of a circular membrane, supported along its outer edge.

GEOMETRY

- Membrane radius, R = 0.25 m
- Membrane thickness h = 0.2 mm

MATERIAL

- Young's modulus, E = 200 GPa
- Poisson's ratio, $\nu = 0.33$
- Mass density, $\rho = 7850\ \mathrm{kg/m^3}$

CONSTRAINTS

The outer edge of the membrane is supported in the transverse direction. Two points have constraints in the in-plane direction in order to avoid rigid body motions.

LOAD

The membrane is pre-tensioned by in the radial direction with σ_i = 100 MPa, giving a membrane force T_0 = 20 kN/m.

Results and Discussion

The analytical solution for the natural frequencies of the vibrating membrane given in Ref. 1 is:

$$f_{ij} = \frac{k_{ij}}{2\pi R}\sqrt{\frac{T_0}{h\rho}} \tag{1}$$

The values k_{ij} are derived from the roots of the Bessel functions of the first kind.

In Table 1 the computed results are compared with the results from Equation 1. The agreement is very good. The mode shapes for the first six modes are shown in Figure 1 through Figure 6. Note that some of the modes have duplicate eigenvalues, which is a common property for structures with symmetries.

TABLE 1: COMPARISON BETWEEN ANALYTICAL AND COMPUTED NATURAL FREQUENCIES

Mode number	**Factor**	**Analytical frequency (Hz)**	**COMSOL result (Hz)**
1	k_{10} = 2.4048	172.8	172.8
2	k_{11} = 3.8317	275.3	275.3
3	k_{11} = 3.8317	275.3	275.3
4	k_{12} = 5.1356	369.0	369.1
5	k_{12} = 5.1356	369.0	369.1
6	k_{20} = 5.5201	396.6	396.7

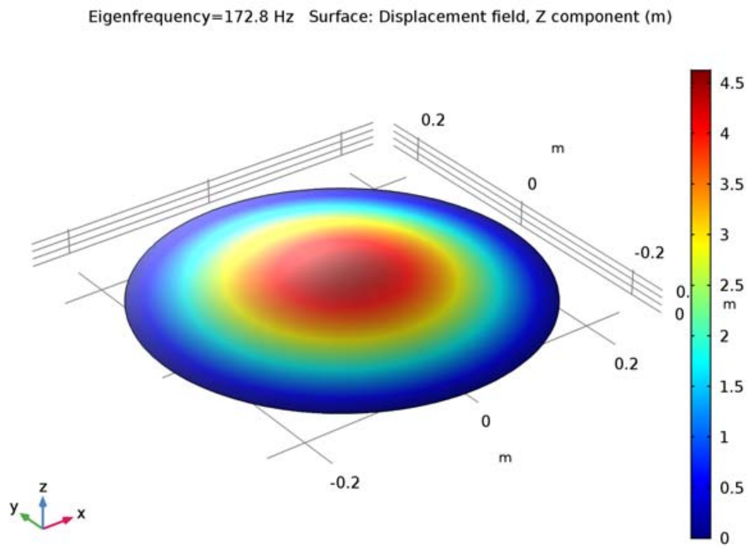


Figure 1: First eigenmode.

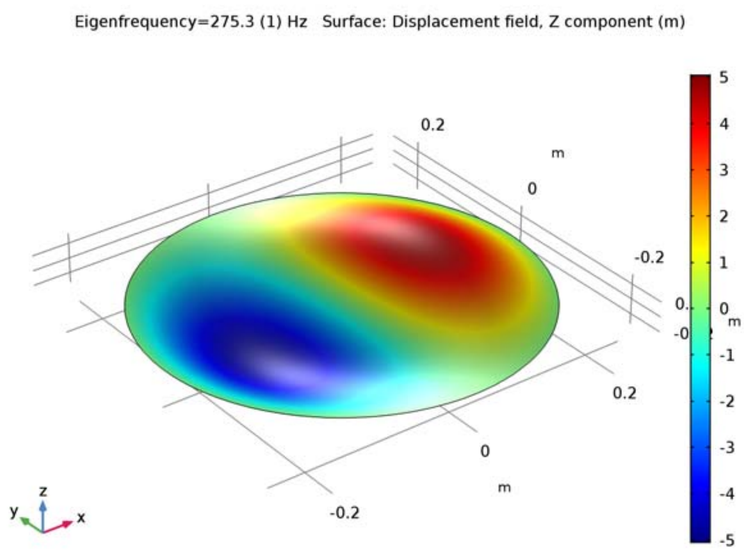


Figure 2: Second eigenmode.

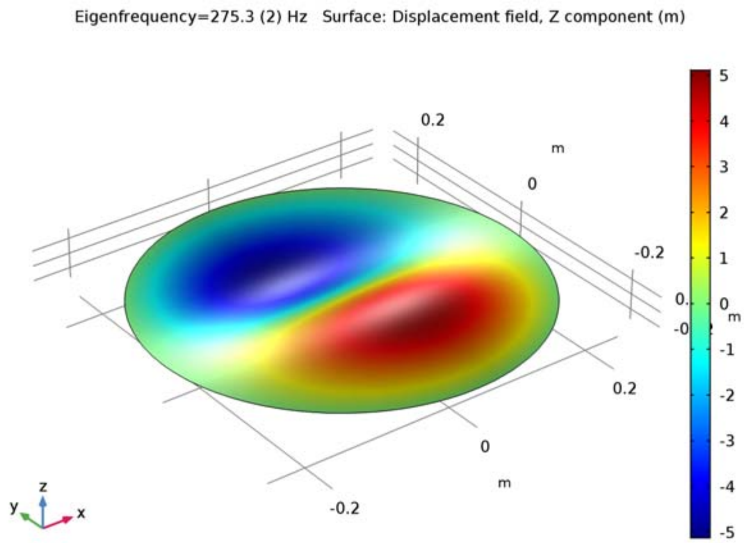


Figure 3: Third eigenmode.

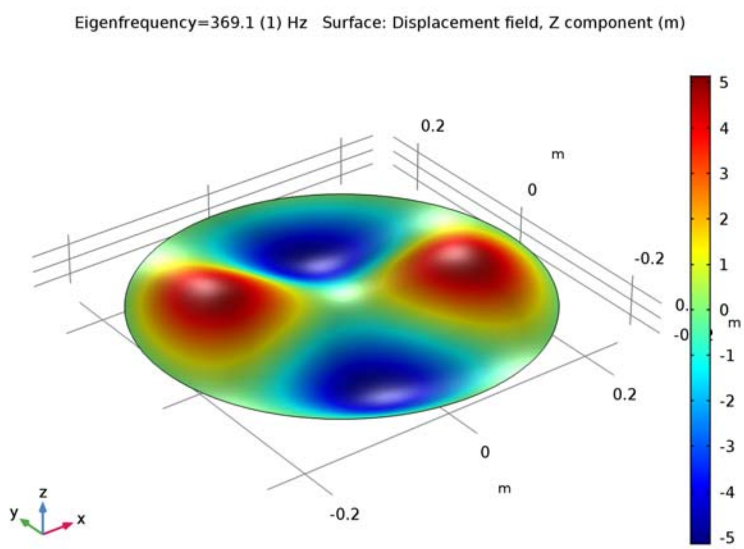


Figure 4: Fourth eigenmode.

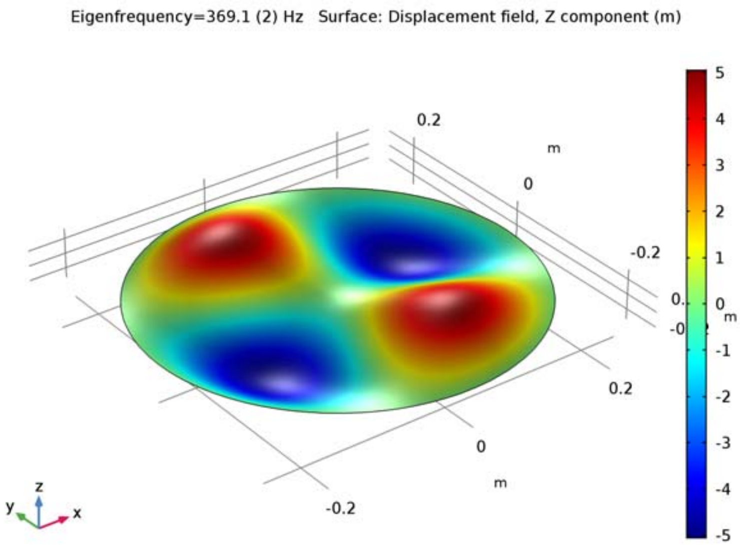


Figure 5: Fifth eigenmode.

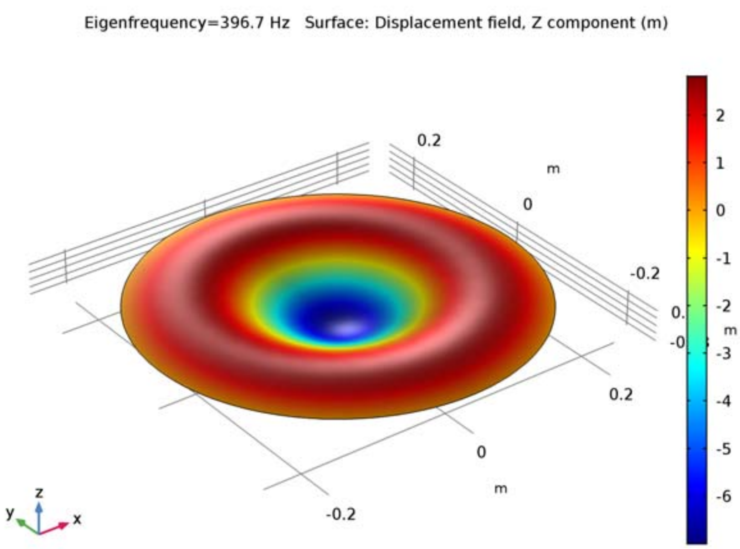


Figure 6: Sixth eigenmode.

Notes About the COMSOL Implementation

An eigenfrequency simulation with a pre-stressed structure can be simulated in two ways. If stresses are known in advance, it is possible to use an initial stress condition. This is shown in the first study.

In a general case, the prestress is given by some external loading, and is thus the result of a previous step in the solution. Such a study would consist of two steps: One stationary step for computing the prestressed state, and one step for the eigenfrequency. The special study type Prestressed Analysis, Eigenfrequency can be used to set up such a sequence. This is shown in the second study in this example.

Since an unstressed membrane has no stiffness in the transverse direction, it is generally difficult to get an analysis to converge without taking special measures. One such method is shown in the second study: A spring foundation is added during initial loading, and is then removed.

Reference

1. A. Bower, *Applied Mechanics of Solids*, CRC Press, 2010.

Application Library path: `Structural_Mechanics_Module/`
`Verification_Examples/vibrating_membrane`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Membrane (mbrn)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Eigenfrequency**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
R	250[mm]	0.25 m	Radius
thic	0.2[mm]	2E-4 m	Thickness
T0	100[MPa]*thic	2E4 N/m	Pre-tension force
E1	200[GPa]	2E11 Pa	Young's modulus
rho1	7850[kg/m^3]	7850 kg/m³	Density
nu1	0.33	0.33	Poisson's ratio
fct	sqrt(T0/(thic* rho1))/(2*pi*R)	71.85 1/s	Common factor in natural frequencies
f10	2.4048*fct	172.8 1/s	1st natural frequency
f11	3.8317*fct	275.3 1/s	2nd and 3d natural frequencies
f12	5.1356*fct	369 1/s	4th and 5th natural frequencies
f20	5.5201*fct	396.6 1/s	6th natural frequency

DEFINITIONS

Cylindrical System 2 (sys2)

On the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

GEOMETRY 1

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `R`.

4 In the **Model Builder** window, click **Geometry 1**.

5 On the **Home** toolbar, click **Build All**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`E1`	Pa	Basic
Poisson's ratio	nu	`nu1`	1	Basic
Density	rho	`rho1`	kg/m³	Basic

MEMBRANE (MBRN)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Membrane (mbrn)**.

2 In the **Settings** window for **Membrane**, locate the **Thickness** section.

3 In the d text field, type `thic`.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Membrane (mbrn)** click **Linear Elastic Material 1**.

Initial Stress and Strain 1

1 On the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

3 In the N_0 table, enter the following settings:

T0	0
0	T0

Prescribed Displacement 1

1 On the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement**.

2 Select all four edges.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in z direction** check box.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 Select Point 1 only.

Prescribed Displacement 2

1 On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.

2 Select Point 2 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in y direction** check box.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Fine**.

STUDY 1

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 1** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 Select the **Include geometric nonlinearity** check box.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Surface

1 In the **Model Builder** window, expand the **Mode Shape (mbrn)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `w`.

4 On the **Mode Shape (mbrn)** toolbar, click **Plot**.

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

Mode Shape (mbrn)

1 In the **Model Builder** window, under **Results** click **Mode Shape (mbrn)**.
2 From the **Eigenfrequency** list, choose the first frequency at **275.3** Hz.
3 On the **Mode Shape (mbrn)** toolbar, click **Plot**.
4 From the **Eigenfrequency** list, choose the first frequency at **275.3** Hz.
5 On the **Mode Shape (mbrn)** toolbar, click **Plot**.
6 From the **Eigenfrequency** list, choose the first frequency at **369.1** Hz.
7 On the **Mode Shape (mbrn)** toolbar, click **Plot**.
8 From the **Eigenfrequency** list, choose the first frequency at **369.1** Hz.
9 On the **Mode Shape (mbrn)** toolbar, click **Plot**.
10 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
11 From the **Eigenfrequency (Hz)** list, choose **396.7**.
12 On the **Mode Shape (mbrn)** toolbar, click **Plot**.

Now, prepare a second study where the prestress is instead computed from an external load.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2 Go to the **Add Study** window.
3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Prestressed Analysis, Eigenfrequency**.
4 Click **Add Study** in the window toolbar.
5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

MEMBRANE (MBRN)

Edge Load 1

1 On the **Physics** toolbar, click **Edges** and choose **Edge Load**.
2 Select all four edges.
3 In the **Settings** window for **Edge Load**, locate the **Coordinate System Selection** section.
4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.
5 Locate the **Force** section. From the **Load type** list, choose **Force per unit length**.

6 Specify the $\mathbf{F}_\mathrm{L}$ vector as

T0	r
0	phi
0	a

Add a spring with an arbitrary small stiffness in order to suppress the out-of-plane singularity of the unstressed membrane.

Spring Foundation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.

2 Click in the **Graphics** window and then press Ctrl+A to select all boundaries.

3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

4 From the **Spring type** list, choose **Spring constant per unit area**.

5 From the list, choose **Diagonal**.

6 In the $\mathbf{k}_\mathrm{A}$ table, enter the following settings:

0	0	0
0	0	0
0	0	10

Switch off the initial stress, which should not be part of the second study. In the eigenfrequency step, the stabilizing spring support must also be removed.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Study Settings** section.

3 Select the **Include geometric nonlinearity** check box.

4 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Membrane (mbrn), Controls spatial frame>Linear Elastic Material 1>Initial Stress and Strain 1**.

6 Click **Disable**.

Step 2: Eigenfrequency

1 In the **Model Builder** window, under **Study 2** click **Step 2: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 Select the **Include geometric nonlinearity** check box.

4 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Linear Elastic Material 1> Initial Stress and Strain 1** and **Component 1 (comp1)>Membrane (mbrn), Controls spatial frame>Spring Foundation 1**.

6 Click **Disable**.

7 On the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (mbrn) 1

The eigenfrequencies computed using this more general approach are the same as before, except some small numerical differences.

To make **Study 1** behave as when it was first created, the features added for **Study 2** must be disabled.

STUDY 1

Step 1: Eigenfrequency

1 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

2 Select the **Modify physics tree and variables for study step** check box.

3 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Edge Load 1** and **Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Spring Foundation 1**.

4 Click **Disable**.

Created in COMSOL Multiphysics 5.3

Vibrating String

Introduction

In the following example you compute the natural frequencies of a pre-tensioned string using the 2D Truss interface. This is an example of "stress stiffening". In fact the transverse stiffness of truss elements is directly proportional to the tensile force.

Strings made of piano wire have an extremely high yield limit, thus enabling a wide range of pre-tension forces.

The results are compared with the analytical solution.

Model Definition

The finite element idealization consists of a single line. The diameter of the wire is irrelevant for the solution of this particular problem, but it must still be given.

GEOMETRY

- String length, $L = 0.5$ m
- Cross section diameter 1.0 mm; $A = 0.785\ \mathrm{mm}^2$

MATERIAL

- Young's modulus, $E = 210$ GPa
- Poisson's ratio, $\nu = 0.31$
- Mass density, $\rho = 7850\ \mathrm{kg/m^3}$

CONSTRAINTS

Both ends of the wire are fixed.

LOAD

The wire is pre-tensioned to $\sigma_{ni} = 1520$ MPa.

Results and Discussion

The analytical solution for the natural frequencies of the vibrating string (Ref. 1) is

$$f_k = \frac{k}{2L}\sqrt{\frac{\sigma_{ni}}{\rho}} \tag{1}$$

The pre-tensioning stress σ_{ni} in this example is tuned so that the first natural frequency is Concert A; 440 Hz.

In Table 1 the computed results are compared with the results from Equation 1. The agreement is very good. The accuracy decreases with increasing complexity of the mode shape, because the possibility for the relatively coarse mesh to describe such a shape is limited. The mode shapes for the first three modes are shown in Figure 1 through Figure 3.

TABLE 1: COMPARISON BETWEEN ANALYTICAL AND COMPUTED NATURAL FREQUENCIES

Mode number	Analytical frequency (Hz)	COMSOL result (Hz)
1	440.0	440.1
2	880.0	880.6
3	1320	1322
4	1760	1765
5	2200	2209

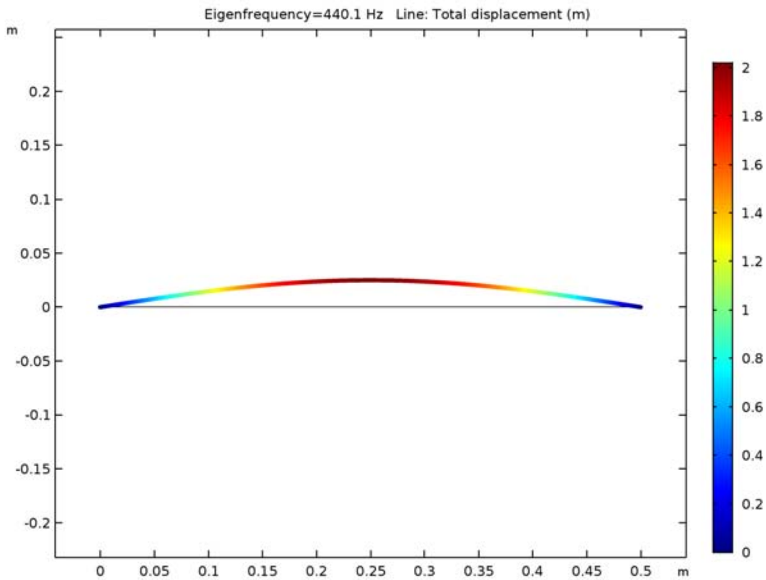


Figure 1: First eigenmode.

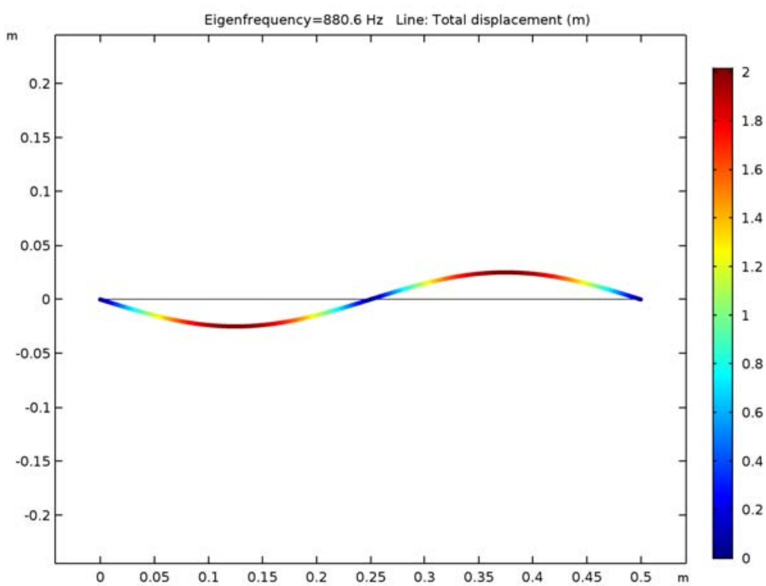


Figure 2: Second eigenmode.

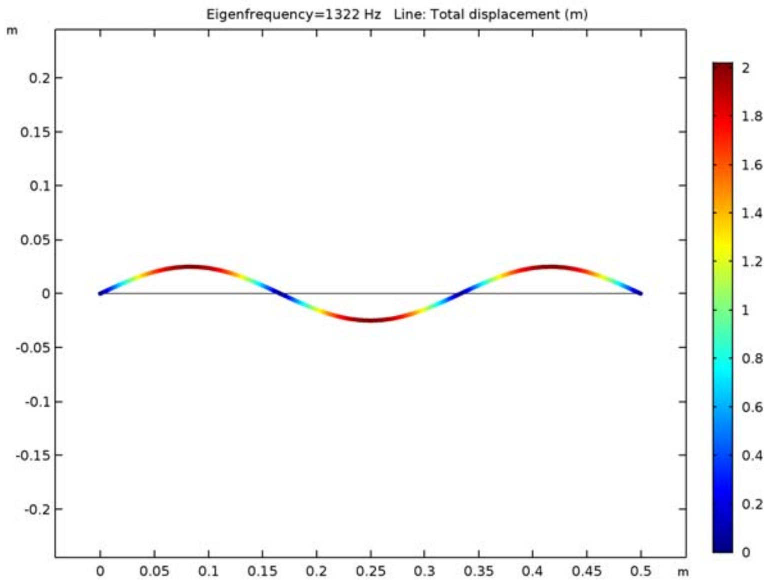


Figure 3: Third eigenmode.

Notes About the COMSOL Implementation

In this example, the stresses are known in advance, so it is possible to use an initial stress condition. This is shown in the first study.

In a general case, the prestress is given by some external loading. The structural response of to this loading needs to be calculated and incorporated into the structure before the eigenfrequency can be computed. Such a study therefore consists of two steps: One stationary step for computing the prestressed state, and one step for the eigenfrequency. The special study type Prestressed Analysis, Eigenfrequency can be used to set up such a sequence. This is shown in the second study in this example.

Since an unstressed membrane has no stiffness in the transverse direction, it is generally difficult to get an analysis to converge without taking special measures. One such method is shown in the second study: A spring foundation is added during initial loading, and is then removed.

You must switch on geometrical nonlinearity in the study in order to capture effects of prestress. This is done automatically when a study of the type Prestressed Analysis, Eigenfrequency is used.

Reference

1. R. Knobel, *An Introduction to the Mathematical Theory of Waves*, The American Mathematical Society, 2000.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/vibrating_string`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Truss (truss)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Eigenfrequency**.

6 Click **Done**.

GEOMETRY 1

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **x** to `0.5`.

5 Click **Build All Objects**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, locate the **Material Contents** section.
3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	210e9	Pa	Basic
Poisson's ratio	nu	0.31	1	Basic
Density	rho	7850	kg/m³	Basic

TRUSS (TRUSS)

Cross Section Data 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Truss (truss)** node, then click **Cross Section Data 1**.
2 In the **Settings** window for **Cross Section Data**, locate the **Cross Section Data** section.
3 In the A text field, type `pi/4*0.001^2`.

Pinned 1

1 In the **Model Builder** window, right-click **Truss (truss)** and choose **Pinned**.
2 In the **Settings** window for **Pinned**, locate the **Point Selection** section.
3 From the **Selection** list, choose **All points**.

The straight edge constraint must be removed because the vibration gives the string a curved shape.

Initial Stress and Strain 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Truss (truss)** right-click **Linear Elastic Material 1** and choose **Initial Stress and Strain**.
2 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.
3 In the σ_{n0} text field, type `1520e6`.

MESH 1

Edge 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Edge**.

2 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.01`.

   This setting gives 50 elements for the mesh that COMSOL Multiphysics generates when you solve the model.

The stiffness caused by the prestress is a nonlinear effect, so geometric nonlinearity must be switched on.

STUDY 1

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 1** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 Select the **Include geometric nonlinearity** check box.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (truss)

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

   The default plot shows the displacement for the first eigenmode.

2 In the **Model Builder** window, under **Results** click **Mode Shape (truss)**.

3 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

4 From the **Eigenfrequency (Hz)** list, choose **880.6**.

   This corresponds to the second eigenmode.

5 On the **Mode Shape (truss)** toolbar, click **Plot**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

7 From the **Eigenfrequency (Hz)** list, choose **1322**.

This is the third eigenmode.

8 On the **Mode Shape (truss)** toolbar, click **Plot**.

9 Click the **Zoom Extents** button on the **Graphics** toolbar.

Now, prepare a second study where the prestress is instead computed from an external load. The pinned condition in the right end must then be replaced by a force.

TRUSS (TRUSS)

Pinned 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Truss (truss)** and choose **Pinned**.

2 Select Point 1 only.

Prescribed Displacement 1

1 In the **Model Builder** window, right-click **Truss (truss)** and choose **Prescribed Displacement**.

2 Select Point 2 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in y direction** check box.

Point Load 1

1 Right-click **Truss (truss)** and choose **Point Load**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_\mathrm{P}$ vector as

1520[MPa]*truss.area	x
0	y

Add a spring with an arbitrary small stiffness in order to suppress the out-of-plane singularity of the unstressed wire.

Spring Foundation 1

1 Right-click **Truss (truss)** and choose the boundary condition **Mass, Spring, and Damper> Spring Foundation**.

2 Click in the **Graphics** window and then press Ctrl+A to select all boundaries.

3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

4 From the list, choose **Diagonal**.

5 In the $\mathbf{k}_L$ table, enter the following settings:

0	0
0	10

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Prestressed Analysis, Eigenfrequency**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

Switch off the initial stress and double-sided pinned condition, which should not be part of the second study. In the eigenfrequency step, the stabilizing spring support must also be removed.

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Truss (truss)> Linear Elastic Material 1>Initial Stress and Strain 1** and **Component 1 (comp1)> Truss (truss)>Pinned 1**.

5 Click **Disable**.

Step 2: Eigenfrequency

1 In the **Model Builder** window, under **Study 2** click **Step 2: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.
3 Select the **Modify physics tree and variables for study step** check box.
4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Truss (truss)>Linear Elastic Material 1>Initial Stress and Strain 1**, **Component 1 (comp1)>Truss (truss)>Pinned 1**, and **Component 1 (comp1)>Truss (truss)>Spring Foundation 1**.
5 Click **Disable**.
6 On the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (truss) 1

The eigenfrequencies computed using this more general approach are close to those computed in the previous step.

To make **Study 1** behave as when it was first created, the features added for **Study 2** must be disabled.

STUDY 1

Step 1: Eigenfrequency

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Eigenfrequency**.
2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.
3 Select the **Modify physics tree and variables for study step** check box.
4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Truss (truss)>Pinned 2**, **Component 1 (comp1)>Truss (truss)>Prescribed Displacement 1**, **Component 1 (comp1)>Truss (truss)>Point Load 1**, and **Component 1 (comp1)>Truss (truss)>Spring Foundation 1**.
5 Click **Disable**.