Structural Mechanics Module

Verification Examples

5.5

COMSOL

Structural Mechanics Module Verification Examples

Version: COMSOL 5.5

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

Contact Information

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.

Block Pressing on Arch

Introduction

This conceptual example shows how to calculate critical points in models with contact. The model consists of a block modeled with the Solid Mechanics interface pressing on an arch modeled with the Shell interface and also exemplifies how to model the contact between a shell and a solid. During loading, the arch exhibits a snap-through behavior. The definition of the problem is based on a benchmark example from Ref. 1.

Model Definition

The model geometry consists of an arch and a block as shown in Figure 1. Since the arch is modeled with the Shell interface, a 3D geometry is used. However, a 2D plane strain behavior is intended, and consequently symmetry conditions are applied to all boundaries and edges in the *y* direction to suppress any out-of-plane deformation.

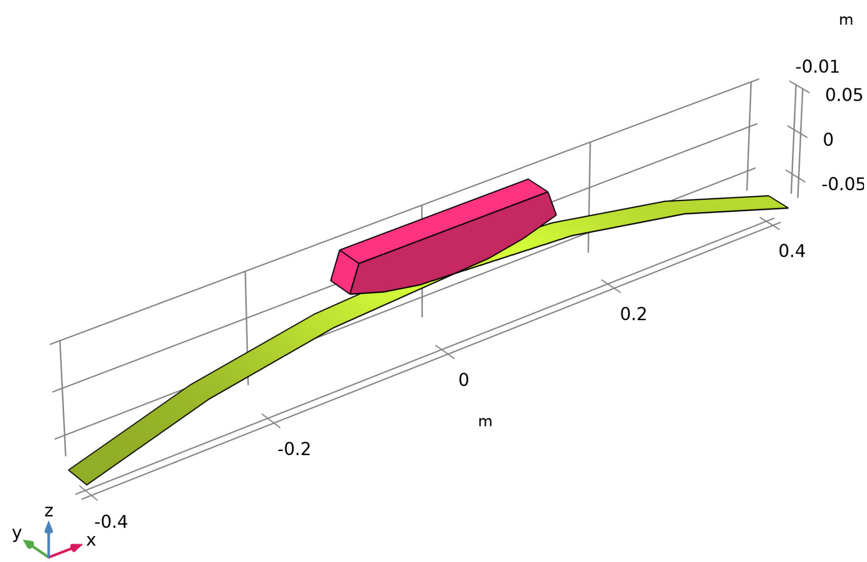


Figure 1: Model geometry

Only contact without friction is considered and the augmented Lagrangian contact method is used.

A boundary load is applied the top surface of the block. Its magnitude is controlled by the monotonically increasing deflection of the arch, which makes it possible to track the entire load path, even though the force does not increase monotonically. The ends of the arch are fixed and the displacement of the block is constrained in the *x* direction.

Results and Discussion

Figure 2 depicts the deformed shape and the von Mises stress distribution at the last step of the simulation. The snap-through of the arch is clearly observed. The arch is represented by a shell dataset that shows both its top and bottom surface.

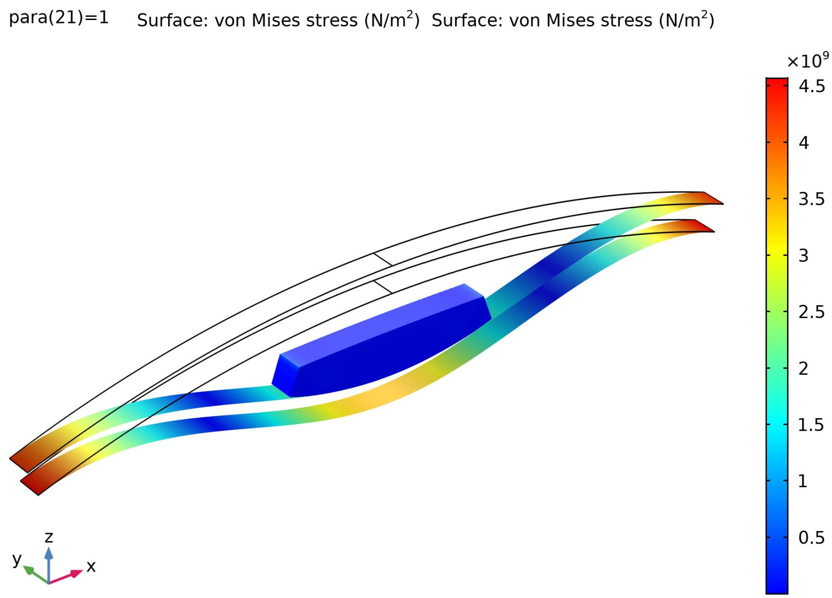


Figure 2: Deformation and von Mises stress at the final step.

The load versus deflection curve is shown in Figure 3. The load is in the figure represented by a dimensionless load factor. Two limit points can be observed, the first occurs for a load factor equal to 18 and a deflection of 36 mm. At this point the arch becomes unstable and a snap-through occurs. When the deflection of the arch reaches 80 mm, the load factor has decreased to 14. At this point the second limit point is reached, and the arch finds a new stable configuration. After this point the load factor increases with increasing deflection.

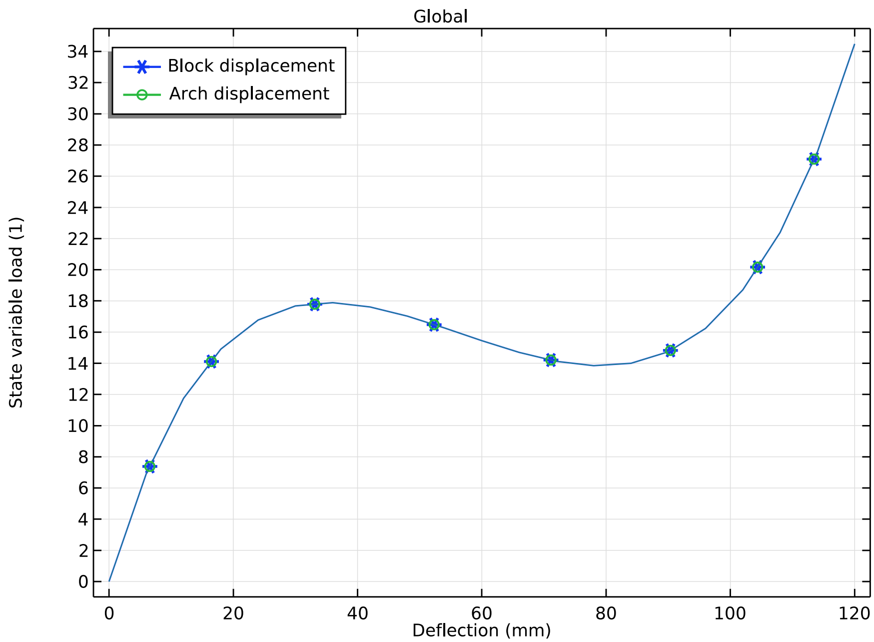


Figure 3: Load versus deflection curve.

The progressive deformation of the block and the arch, including the snap-through of the arch, is shown in Figure 4 for six values of the continuation parameter. Figure 5 shows the contact pressure exerted by the block on the arch during the snap-through.

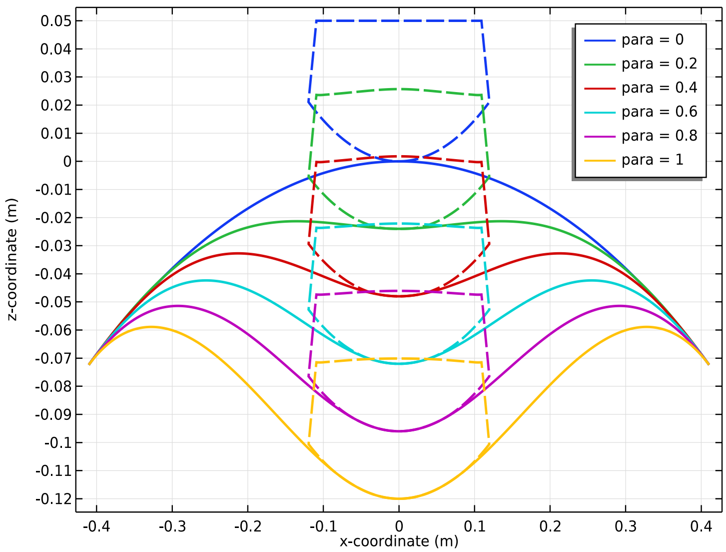


Figure 4: Deformation of the model for six different parameter values.

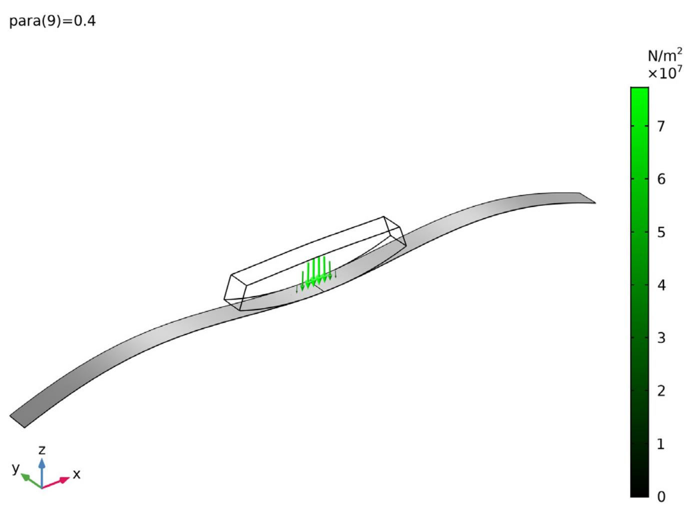


Figure 5: Contact pressure acting on the arch.

Notes About the COMSOL Implementation

When a Shell interface is used in a contact simulation, it is recommended that the destination boundary always belongs to the shell. Moreover, the contact definition should be made in the Shell interface. In this example, the block modeled with a Solid Mechanics interface is thus, in the **Contact** node, considered as external to the current physics.

Contact problems are often unstable in their initial configuration. To help the solver find an initial solution, a **Spring Foundation** is added to the otherwise unconstrained block during the first parameter step.

Modeling the post-critical behavior of a system is not possible by incrementally increasing the boundary load. The unstable behavior is even more pronounced when contact is present. To be able to find all limit points and to track the full load versus deflection curve, a displacement controlled load scheme is used by adding a **Global Equation**. Here, the magnitude of the boundary load is controlled through the monotonically increasing deflection of the arch. Alternatively, the vertical displacement could be prescribed on the top surface of the block, but this is a less general technique that fails for some cases. Also, a prescribed displacement would not give an evenly distributed load.

Reference

1. P. Wriggers, *Computational Contact Mechanics*, Springer-Verlag, 2006

Application Library path: `Structural_Mechanics_Module/Verification_Examples/block_on_arch`

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>Solid Mechanics (solid)**.

5 Click **Add**.

6 In the **Displacement field** text field, type `u`.

7 Click **Study**.

8 In the **Select Study** tree, select **General Studies>Stationary**.

9 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `block_on_arch_parameters.txt`.

GEOMETRY 1

Work Plane 1 (wp1)

1 In 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**.

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `R_arch`.

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

6 Locate the **Position** section. In the **yw** text field, type `-R_arch`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `90-seg_arch/2`.

8 Click **Build Selected**.

9 Click the **Zoom Extents** button in the **Graphics** toolbar.

Work Plane 1 (wp1)>Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Plane Geometry** and choose **Delete Entities**.

2 On the object **c1**, select Boundaries 2 and 3 only.

Work Plane 1 (wp1)>Partition Edges 1 (pare1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Partition Edges**.

2 On the object **del1**, select Boundary 1 only.

Work Plane 1 (wp1)>Circle 2 (c2)

1 In the **Work Plane** toolbar, click **Circle**.

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

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

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

5 Locate the **Position** section. In the **yw** text field, type `R_block`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90-seg_block/2`.

7 Click **Build Selected**.

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

Work Plane 1 (wp1)>Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type `R_block`.

4 In the **Height** text field, type `height_block`.

5 Locate the **Position** section. In the **xw** text field, type `-R_block/2`.

6 Click **Build Selected**.

Work Plane 1 (wp1)>Intersection 1 (int1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Intersection**.

2 Select the objects **c2** and **r1** only.

Work Plane 1 (wp1)

1 In the **Model Builder** window, click **Work Plane 1 (wp1)**.

2 In the **Settings** window for **Work Plane**, locate the **Unite Objects** section.

3 Clear the **Unite objects** check box.

Extrude 1 (ext1)

1 In 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)
d

4 Click **Build Selected**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Explicit Selection 1 (sel1)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Arch` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Object**.

4 Select the object **ext1(1)** only.

5 Locate the **Color** section. From the **Color** list, choose **Color 4**.

6 Click **Build Selected**.

Arch 1 (sel2)

1 Right-click **Arch** and choose **Duplicate**.

2 In the **Settings** window for **Explicit Selection**, type `Block` in the **Label** text field.

3 Locate the **Entities to Select** section. In the list, select **ext1(1)**.

4 Select the object **ext1(2)** only.

5 Locate the **Color** section. From the **Color** list, choose **Color 12**.

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 Click **Build Selected**.

5 Click the **Zoom Extents** button in 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	Variable	Value	Unit	Property group
Young's modulus	E	10[GPa]	Pa	Basic
Poisson's ratio	nu	0.2	1	Basic
Density	rho	1	kg/m³	Basic

Material 2 (mat2)

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

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

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

4 From the **Selection** list, choose **Arch**.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	70[GPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	1	kg/m³	Basic

DEFINITIONS

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

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

4 Select Point 11 only.

Average 2 (aveop2)

1 Right-click **Average 1 (aveop1)** and choose **Duplicate**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

3 Click **Clear Selection**.

4 Select Point 3 only.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
disp_block	aveop1(-w)	m	Block displacement
disp_arch	aveop2(-w)	m	Arch displacement

Contact Pair 1 (p1)

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

2 Select Boundaries 4 and 8 only.

3 Click the **Go to Default View** button in the **Graphics** toolbar.

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

5 From the **Selection** list, choose **Arch**.

The destination boundary should be on a boundary modeled with the Shell interface.

SHELL (SHELL)

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

2 In the **Settings** window for **Shell**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Arch**.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

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

4 From the **Offset definition** list, choose **Relative offset**.

5 In the $z_{\text{reloffset}}$ text field, type `-1`.

Prescribed Displacement/Rotation 1

1 In the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement/Rotation**.

2 Select Edges 1 and 7 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 z direction** check box.

6 Locate the **Prescribed Rotation** section. From the **By** list, choose **Rotation**.

Symmetry 1

1 In the **Physics** toolbar, click **Edges** and choose **Symmetry**.

2 Select Edges 2, 3, 5, and 6 only.

Contact 1

1 In 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 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (p1)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Contact**, locate the **Contact Method** section.

7 From the **Formulation** list, choose **Augmented Lagrangian**.

8 Select the **Source external to current physics** check box.

The source boundary is in the Solid Mechanics interface.

SOLID MECHANICS (SOLID)

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

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement**.

2 Select Edges 13 and 19 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

Symmetry 1

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

2 Select Boundaries 5 and 6 only.

Boundary Load 1

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

2 Select Boundary 7 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
load*F_ref	z

The dependent variable `load` will be created in the next step using a global equation.

5 Click the **Show More Options** button in the **Model Builder** toolbar.

6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

7 Click **OK**.

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **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
load	disp_block-para*max_disp	0	0	

4 Locate the **Units** section. Click **Select Source Term Quantity**.

5 In the **Physical Quantity** dialog box, type `displacement` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Displacement (m)**.

8 Click **OK**.

Add a small spring stiffness to the block to stabilize the model during the initial step.

Spring Foundation 1

1 In the **Physics** toolbar, click **Domains** and choose **Spring Foundation**.

2 In the **Settings** window for **Spring Foundation**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Block**.

4 Locate the **Spring** section. In the $\mathbf{k}_V$ text field, type `1e3*(para<0.01)`.

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 **Arch**.

Distribution 1

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

Mapped 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Mapped**.
2 Select Boundary 5 only.

Distribution 1

1 Right-click **Mapped 2** and choose **Distribution**.
2 Select Edges 10 and 17 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type `n_elem_block`.

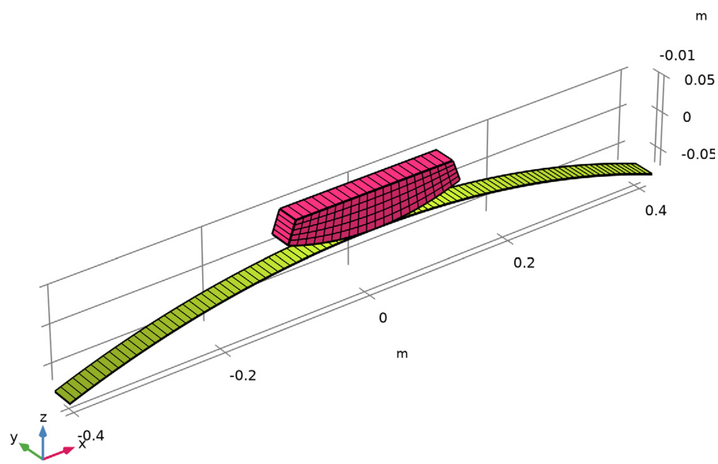
Distribution 2

1 In the **Model Builder** window, right-click **Mapped 2** and choose **Distribution**.
2 Select Edges 9 and 20 only.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.
2 In the **Settings** window for **Mesh**, click **Build All**.

3 Click the **Zoom Extents** button in the **Graphics** toolbar.



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. 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 (Load parameter)	`range(0,0.05,1)`	

Solution 1 (sol1)

In the **Study** toolbar, click **Show Default Solver**.

Stationary Solver 1

1. In the **Model Builder** window, expand the **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 `0.0005`.

State variable load (comp1.ODE1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 1** node, then click **State variable load (comp1.ODE1)**.
2 In the **Settings** window for **State**, locate the **Scaling** section.
3 From the **Method** list, choose **Manual**.

Segregated 1

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

Shell

1 In the **Model Builder** window, expand the **Segregated 1** node, then click **Shell**.
2 In the **Settings** window for **Segregated Step**, locate the **General** section.
3 Under **Variables**, click **Add**.
4 In the **Add** dialog box, select **State variable load (comp1.ODE1)** in the **Variables** list.
5 Click **OK**.

Solid Mechanics

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose **Delete**.

   Structural mechanics interfaces should be solved in a single segregated step.
2 In the **Study** toolbar, click **Compute**.

RESULTS

Surface 2

1 In the **Model Builder** window, expand the **Stress (shell)** node.
2 Right-click **Results>Stress (shell)>Surface 1** and choose **Duplicate**.
3 In the **Settings** window for **Surface**, locate the **Data** section.
4 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
5 Locate the **Expression** section. In the **Expression** text field, type `solid.mises`.
6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.
7 In the **Stress (shell)** toolbar, click **Plot**.
8 Click the **Show Grid** button in the **Graphics** toolbar.
9 Click the **Zoom Extents** button in the **Graphics** toolbar.

Contact (shell)

1 In the **Model Builder** window, click **Contact (shell)**.

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

3 From the **Parameter value (para)** list, choose **0.4**.

Contact 1, Pressure

1 In the **Model Builder** window, expand the **Contact (shell)** node, then click **Contact 1, Pressure**.

2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.

3 Select the **Scale factor** check box.

4 In the associated text field, type `5e-10`.

Gray Surfaces

In the **Model Builder** window, right-click **Gray Surfaces** and choose **Enable**.

Selection 1

1 In the **Model Builder** window, expand the **Gray Surfaces** node.

2 Right-click **Gray Surfaces** and choose **Selection**.

3 Select Boundary 1 only.

4 In the **Settings** window for **Selection**, locate the **Selection** section.

5 From the **Selection** list, choose **Arch**.

6 In the **Contact (shell)** toolbar, click **Plot**.

Animation 1

1 In the **Contact (shell)** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Frames** section.

3 From the **Frame selection** list, choose **All**.

4 Right-click **Animation 1** and choose **Play**.

1D Plot Group 7

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

2 In the **Settings** window for **1D Plot Group**, type `Load vs Deflection` in the **Label** text field.

Global 1

1 Right-click **Load vs Deflection** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`disp_block`	`mm`	`Block displacement`
`disp_arch`	`mm`	`Arch displacement`

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

5 In the **Expression** text field, type `load`.

6 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

Load vs Deflection

1 In the **Model Builder** window, click **Load vs Deflection**.

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

3 Select the **Flip the x- and y-axes** check box.

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Deflection (mm)`.

7 In the **Load vs Deflection** toolbar, click **Plot**.

1D Plot Group 8

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

2 In the **Settings** window for **1D Plot Group**, type `Deformation` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (para)** list, choose **Manual**.

4 In the **Parameter indices (1-21)** text field, type `range(1,4,21)`.

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

Line Graph 1

1 Right-click **Deformation** and choose **Line Graph**.

2 Select Edges 2 and 5 only.

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

4 In the **Expression** text field, type `z`.

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

6 In the **Expression** text field, type `x`.

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

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 Select the **Activate selection** toggle button.

4 Select Edges 9, 10, 14, 17, and 20 only.

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

6 From the **Color** list, choose **Cycle (reset)**.

Line Graph 1

1 In the **Model Builder** window, click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 Find the **Include** subsection. In the **Prefix** text field, type `para = `.

5 In the **Deformation** toolbar, click **Plot**.

Stress (shell)

Click the **Zoom Extents** button in the **Graphics** toolbar.

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.

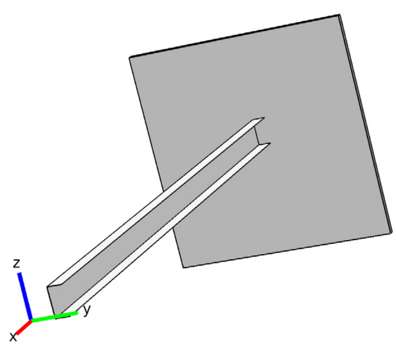


Figure 1: The physical geometry.

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.

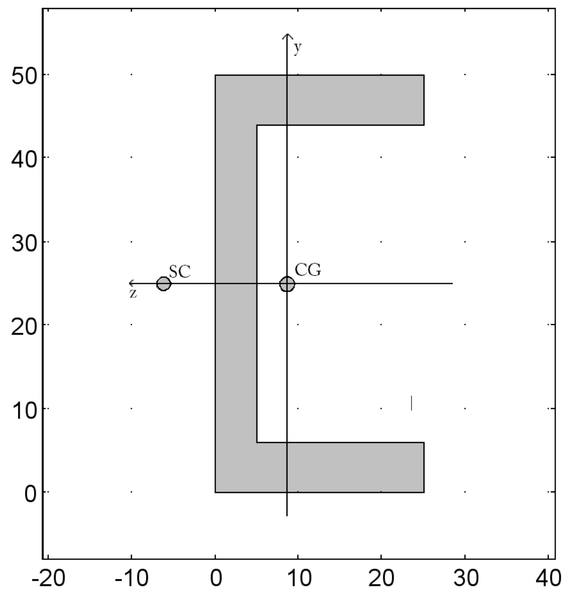


Figure 2: The beam cross section with local direction indicated.

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, –0.0164)
  (y_2, z_2)=(0.025, –0.0164)
  (y_3, z_3)=(0.025, 0.0086),
  (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, $\rho = 7800$ 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

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

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

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

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

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

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

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

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

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

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

$$\sigma_{x, Mz} = \frac{-M_z y}{I_{zz}} = \frac{-F_y L y}{I_{zz}} = \frac{-F_Z L y}{I_{zz}} = \tag{1}$$

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

$$\sigma_{x, My} = \frac{M_y z}{I_{yy}} = \frac{-F_z L z}{I_{yy}} = \frac{F_Y L z}{I_{yy}} = \tag{2}$$

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

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 (=F_X)	Stress from F_y (=-F_Z)	Stress from F_z (=F_Y)	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

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

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

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

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

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 equivalent 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:

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

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

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

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

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

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.

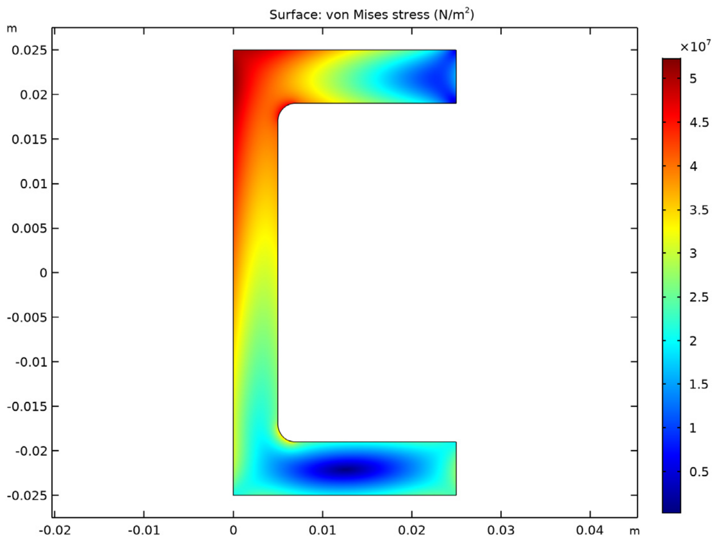


Figure 3: von Mises stress distribution at the fixed end (x = 0).

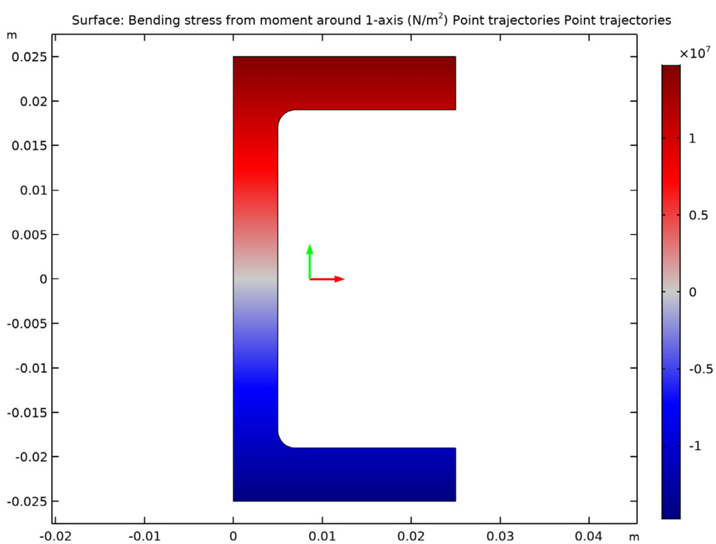


Figure 4: Plot of stresses from a bending moment. The center of gravity is highlighted.

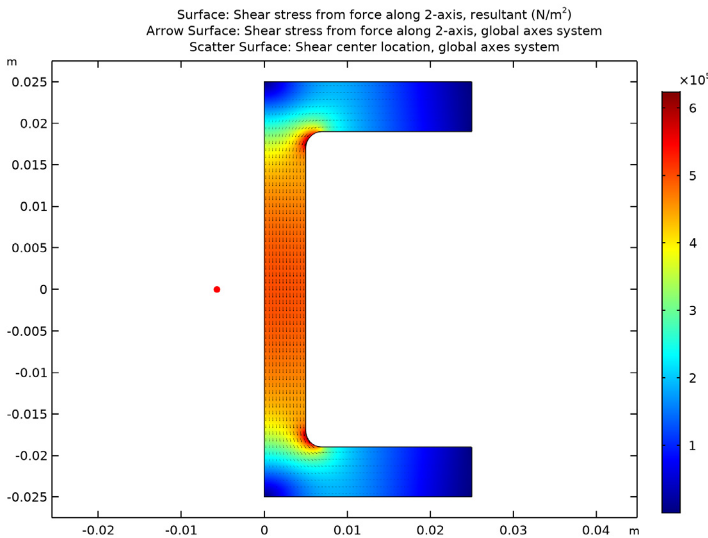


Figure 5: Plot of stresses from shear force. The shear center is highlighted.

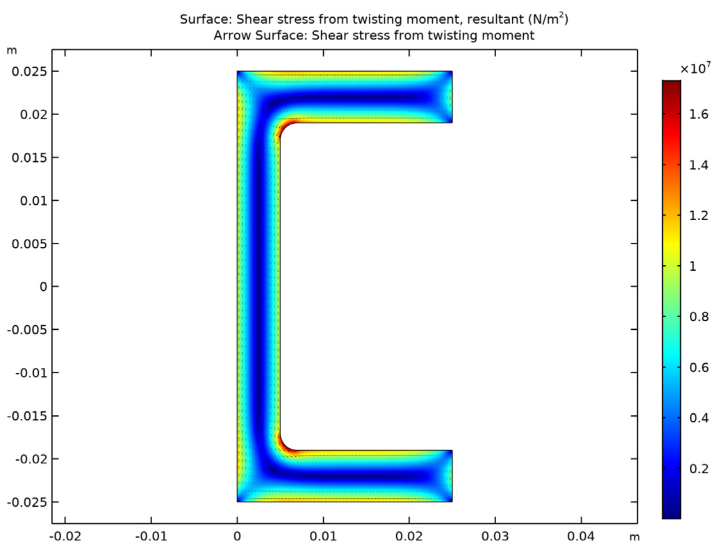


Figure 6: Plot of shear stresses from torsion.

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 **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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`
`L`	`1[m]`	1 m	`Beam length`
`Eb`	`2e11[Pa]`	2E11 Pa	`Young's modulus`
`nub`	`0.25`	0.25	`Poisson's ratio`
`rhob`	`8000[kg/m^3]`	8000 kg/m³	`Density`
`FX`	`10e3[N]`	10000 N	`Force in X direction`
`FY`	`50[N]`	50 N	`Force in Y direction`
`FZ`	`100[N]`	100 N	`Force in Z direction`
`MX`	`-10[N*m]`	-10 N·m	`Moment in X direction`

Load Group 1

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Load and Constraint Groups>Load Group**.

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

Load Group 2

1 In the **Model Builder** window, right-click **Load and Constraint Groups** and choose **Load Group**.

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

GEOMETRY 1

Polygon 1 (pol1)

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

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

3 In the table, enter the following settings:

x (m)	y (m)	z (m)
0	0	0
1	0	0

4 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	Variable	Value	Unit	Property group
Young's modulus	E	Eb	Pa	Basic
Poisson's ratio	nu	nub	1	Basic
Density	rho	rhob	kg/m³	Basic

DEFINITIONS

Define the cross section parameters to compute the analytical values of the displacement and section forces of the beam.

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Gb	Eb/(2*(1+nub))	Pa	Shear Modulus
A	4.9e-4[m^2]	m²	Cross section area
Iyy	2.77e-8[m^4]	m^4	Area moment of inertia, y component
Izz	1.69e-7[m^4]	m^4	Area moment of inertia, z component

Name	Expression	Unit	Description
Jbeam	5.18e-9[m^4]	m^4	Torsion constant
Wt	8.64e-7[m^3]	m³	Torsion section modulus
ey	0[m]	m	Shear center relative to centroid, y-coordinate
ez	0.0148[m]	m	Shear center relative to centroid, z-coordinate
muy	2.44		Max shear stress factor in local y direction
muz	2.38		Maximum shear stress factor in local z direction
y1	-0.025[m]	m	Evaluation point 1, local y-coordinate
z1	-0.0164[m]	m	Evaluation point 1, local z-coordinate
y2	0.025[m]	m	Evaluation point 2, local y-coordinate
z2	-0.0164[m]	m	Evaluation point 2, local z-coordinate
y3	0.025[m]	m	Evaluation point 3, local y-coordinate
z3	0.0086[m]	m	Evaluation point 3, local z-coordinate
y4	-0.025[m]	m	Evaluation point 4, local y-coordinate
z4	0.0086[m]	m	Evaluation point 4, local z-coordinate

Define an analytic function to evaluate the bending stress at different locations of the cross section.

Analytic 1 (an1)

1 In the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `sigmabx` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `-FZ*L*y/comp1.Izz+FY*L*z/comp1.Iyy`.

4 In the **Arguments** text field, type `y, z`.

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

Argument	Lower limit	Upper limit
y	-h2/2	h2/2
z	-h1/2	h1/2

6 Locate the **Units** section. In the **Arguments** text field, type `m, m`.

7 In the **Function** text field, type `N/m^2`.

8 Right-click **Analytic 1 (an1)** and choose **Rename**.

9 In the **Rename Analytic** dialog box, type `sigmabx` in the **New label** text field.

10 Click **OK**.

Define the variables for analytical values of the displacements, rotations and stresses.

Variables 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
deltaX	FX*L/(Eb*A)	m	X displacement
deltaY	FY*L^3/(3*Eb*Iyy)	m	Y displacement
deltaZ	FZ*L^3/(3*Eb*Izz)	m	Z displacement
thetaX	MX*L/(Gb*Jbeam)		Twist
sigmax_Fx	FX/A	N/m²	Stress due to axial load
tausy_max	muy*FZ/A	N/m²	Maximum shear stress due y force
tausz_max	-muz*FY/A	N/m²	Maximum shear stress due to z force
taut_max	abs(MX)/Wt	N/m²	Shear stress due to torsion
tauxz_max	abs(tausz_max)+taut_max	N/m²	Maximum shear stress, z component

Name	Expression	Unit	Description
tauxy_max	abs(tausy_max)+taut_max	N/m²	Maximum shear stress, y component
sigx1	sigmax_Fx+sigmabx(y1,z1)	N/m²	Normal stress at point 1
sigx2	sigmax_Fx+sigmabx(y2,z2)	N/m²	Normal stress at point 2
sigx3	sigmax_Fx+sigmabx(y3,z3)	N/m²	Normal stress at point 3
sigx4	sigmax_Fx+sigmabx(y4,z4)	N/m²	Normal stress at point 4
sigx_max	max(max(max(sigx1,sigx2), sigx3),sigx4)	N/m²	Maximum normal stress in cross section
sig_mises	sqrt(sigx_max^2+3*tauxy_max^2+ 3*tauxz_max^2)	N/m²	Maximum von Mises stress
deltaZ_g	-rhob*g_const*A*L^4/(8*Eb*Izz)	m	Z displacement due to gravity load
thetaX_g	rhob*g_const*A*ez*L^2/(2*Gb* Jbeam)		Twist due to gravity load

BEAM (BEAM)

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. 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. In the **Physics** toolbar, click **Edges** and choose **Gravity**.
2. Select Edge 1 only.
3. In the **Physics** toolbar, click **Load Group** and choose **Load Group 1**.

Fixed Constraint 1

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

Point Load 1

1. In 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

FX	x
FY	y
FZ	z

5. Locate the **Moment** section. Specify the $\mathbf{M}_\mathrm{P}$ vector as

MX	x
0	y
0	z

6. In 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 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 In the **Model Builder** window, right-click **Study 1** and choose **Rename**.

7 In the **Rename Study** dialog box, type `Stationary Study: Beam` in the **New label** text field.

8 Click **OK**.

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

2 From the **Load case** list, choose **Point load**.

3 In 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 In the **Results** toolbar, click **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, type `Case1: Displacement/Rotation` in the **Label** text field.

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

4 Select Point 2 only.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Displacement>Displacement field - m>u - Displacement field, x component**.

6 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Variables>deltaX - X displacement - m**.

7 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Displacement>Displacement field - m>v - Displacement field, y component**.

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Variables>deltaY - Y displacement - m**.

9 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Displacement>Displacement field - m>w - Displacement field, z component**.

10 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Variables>deltaZ - Z displacement - m**.

11 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Displacement>Rotation field - rad>thx - Rotation field, X component**.

12 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Variables>thetaX - Twist**.

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

Expression	Unit	Description
u	m	delta_x computed
deltaX	m	delta_x analytical
v	m	delta_y computed
deltaY	m	delta_y analytical
w	m	delta_z computed
deltaZ	m	delta_z analytical
thx	rad	theta_x computed
thetaX	1	theta_x analytical

14 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 `Case1: Displacement/Rotation` in the **Label** text field.

Point Evaluation 2

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

2 In the **Settings** window for **Point Evaluation**, type `Case2: Displacement/Rotation` in the **Label** text field.

3 Select Point 2 only.

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

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Displacement>Displacement field - m>w - Displacement field, z component**.

6 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Variables>deltaZ_g - Z displacement due to gravity load - m**.

7 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Displacement>Rotation field - rad>thx - Rotation field, X component**.

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Variables>thetaX_g - Twist due to gravity load**.

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

Expression	Unit	Description
`w`	`m`	`delta_z computed`
`deltaZ_g`	`m`	`delta_z analytical`
`thx`	`rad`	`theta_x computed`
`thetaX_g`	`1`	`theta_x analytical`

10 Click **Evaluate**.

Table 2

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

2 In the **Settings** window for **Table**, type `Case2: Displacement/Rotation` in the **Label** text field.

Point Evaluation 3

1 In 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 **Model>Component 1>Beam>Stress> Stress variables at first evaluation point>beam.s1 - Normal stress at first evaluation point - N/m²**.

7 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Stress> Stress variables at second evaluation point>beam.s2 - Normal stress at second evaluation point - N/m²**.

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Stress> Stress variables at third evaluation point>beam.s3 - Normal stress at third evaluation point - N/m²**.

9 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Stress> Stress variables at fourth evaluation point>beam.s4 - Normal stress at fourth evaluation point - N/m²**.

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 3

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

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

Point Evaluation 4

1 In 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 **Model>Component 1>Beam>Stress>Stress variables at first evaluation point>beam.sb1 - Bending stress at first evaluation point - N/m²**.

6 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Functions>sigmabx(y, z) - sigmabx**.

7 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Stress>Stress variables at second evaluation point>beam.sb2 - Bending stress at second evaluation point - N/m²**.

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Functions>sigmabx(y, z) - sigmabx**.

9 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Stress>Stress variables at third evaluation point>beam.sb3 - Bending stress at third evaluation point - N/m²**.

10 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Functions>sigmabx(y, z) - sigmabx**.

11 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Stress>Stress variables at fourth evaluation point>beam.sb4 - Bending stress at fourth evaluation point - N/m²**.

12 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Functions>sigmabx(y, z) - sigmabx**.

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

Expression	Unit	Description
`beam.sb1`	`MPa`	`first point, computed`
`sigmabx(y1, z1)`	`MPa`	`first point, analytical`
`beam.sb2`	`MPa`	`second point, computed`
`sigmabx(y2, z2)`	`MPa`	`second point, analytical`
`beam.sb3`	`MPa`	`third point, computed`

Expression	Unit	Description
sigmabx(y3, z3)	MPa	third point, analytical
beam.sb4	MPa	fourth point, computed
sigmabx(y4, z4)	MPa	fourth point, analytical

14 Click **Evaluate**.

Table 4

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

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

Point Evaluation 5

1 In 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 **Model>Component 1>Beam>Stress>beam.tsymax - Max shear stress from shear force, y direction - N/m²**.

6 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Variables>tausy_max - Maximum shear stress due y force - N/m²**.

7 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Stress>beam.tszmax - Max shear stress from shear force, z direction - N/m²**.

8 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Variables>tausz_max - Maximum shear stress due to z force - N/m²**.

9 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Stress>beam.ttmax - Max torsional shear stress - N/m²**.

10 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Variables>taut_max - Shear stress due to torsion - N/m²**.

11 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Stress>beam.txymax - Max shear stress, y direction - N/m²**.

12 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Variables>tauxy_max - Maximum shear stress, y component - N/m²**.

13 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Beam>Stress>beam.txzmax - Max shear stress, z direction - N/m²**.

14 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Definitions>Variables>tauxz_max - Maximum shear stress, z component - N/m²**.

15 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 (Computed)`
`tausy_max`	`MPa`	`Max shear stress from shear force, y direction (Analytical)`
`beam.tszmax`	`MPa`	`Max shear stress from shear force, z direction (Computed)`
`tausz_max`	`MPa`	`Max shear stress from shear force, z direction (Analytical)`
`beam.ttmax`	`MPa`	`Max torsional shear stress (Computed)`
`taut_max`	`MPa`	`Max torsional shear stress (Analytical)`
`beam.txymax`	`MPa`	`Max shear stress, y direction (Computed)`
`tauxy_max`	`MPa`	`Max shear stress, y direction (Analytical)`
`beam.txzmax`	`MPa`	`Max shear stress, z direction (Computed)`
`tauxz_max`	`MPa`	`Max shear stress, z direction (Analytical)`

16 Click **Evaluate**.

Table 5

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

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

Perform an eigenfrequency analysis.

ADD STUDY

1 In 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 **General Studies> Eigenfrequency**.
4 Click **Add Study** in the window toolbar.
5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.
2 In the **Rename Study** dialog box, type `Eigenfrequency Study: Beam` in the **New label** text field.
3 Click **OK**.

Step 1: Eigenfrequency

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

1 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
2 Select the **Desired number of eigenfrequencies** check box.
3 In the associated text field, type `20`.
4 In the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (beam)

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
2 From the **Eigenfrequency (Hz)** list, choose **51.956**.
3 In 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.

Datasets

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. To make it possible to change this location we start by creating a **Cut Point**.

Cut Point 3D 1

1 In 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`.
4 In the **Y** text field, type `0`.

5 In the **Z** text field, type `0`.

Point Evaluation 6

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.

4 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

5 Click **Evaluate**.

Table 6

1 In the **Model Builder** window, under **Results>Tables** click **Table 6**.

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

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>2D**.

ADD PHYSICS

1 In 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 2** in the window toolbar.

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

ROOT

In the **Model Builder** window, click the root node.

ADD STUDY

1 In 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 **General 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 From the **Home** menu, choose **Add Study**.

COMPONENT 2 (COMP2)

1 In the **Model Builder** window, collapse the **Component 2 (comp2)** node.

2 Right-click **Study 3** and choose **Rename**.

3 In the **Rename Study** dialog box, type `Stationary Study: Beam Cross Section` in the **New label** text field.

4 Click **OK**.

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

GEOMETRY 2

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

PART LIBRARIES

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.

2 In the **Part Libraries** window, select **Structural Mechanics Module>Beams>Generic>C_beam_generic** in the tree.

3 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
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, click **Form Union (fin)**.

2 Click **Build Selected**.

3 Click the **Zoom Extents** button in the **Graphics** toolbar.

BEAM CROSS SECTION (BCS)

Input the section force data evaluated previously from the **Beam** into **Beam Cross Section**. To automate this process of transferring the section forces at any arbitrary location, create a model method first.

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

NEW METHOD

1 In the **Developer** toolbar, click **New Method**.

2 In the **New Method** dialog box, type `EvaluateSectionForces` in the **Name** text field.

3 Click **OK**.

APPLICATION BUILDER

1 Copy the following code into the **EvaluateSectionForces** window:

```
double Len = model.param().evaluate("L");
String xPos = xp;
try {
  double xP = Double.valueOf(xp);
  if (xP < 0) {
    alert("Evaluation point out of range. Using the root of the beam for
evaluation.", "Evaluation point out of range warning");
    xPos  = "0"
  }
```

```
    if (xP > Len) {
      alert("Evaluation point out of range. Using the tip of the beam for 
  evaluation.", "Evaluation point out of range warning");
      xPos = "L";
    }
  } catch (Exception e) {

  }

  with(model.result().dataset("cpt1"));
    set("pointx", xPos);
  endwith();

  double[][] SecForce = model.result().numerical("pev6").getReal();
  with(model.component("comp2").physics("bcs").prop("UserInput"));
    set("N", Double.toString(SecForce[0][0]));
    set("M1", Double.toString(SecForce[1][0]));
    set("T2", Double.toString(SecForce[2][0]));
    set("M2", Double.toString(SecForce[3][0]));
    set("T1", Double.toString(SecForce[4][0]));
    set("Mt", Double.toString(SecForce[5][0]));
  endwith();
```

2 In the **Application Builder** window, under **Methods** click **EvaluateSectionForces**.

3 In the **Settings** window for **Method**, locate the **Inputs and Output** section.

4 Click **Add**.

5 Find the **Inputs** subsection. In the table, enter the following settings:

Name	Type	Default	Description	Unit
xp	String	0		

METHODS

EvaluateSectionForces

In the **Home** toolbar, click **Model Builder** to switch to the main desktop.

GLOBAL DEFINITIONS

EvaluateSectionForces 1

1 In the **Home** toolbar, click **Method Call** and choose **EvaluateSectionForces**.

Run the method **EvaluateSectionForces** to transfer the cross section forces in **Beam Cross Section** interface.

2 Click **Run Method Call** and choose **EvaluateSectionForces 1**.

STATIONARY STUDY: BEAM CROSS SECTION

Click **Compute**.

Evaluate the beam physical properties required for the **Beam** interface.

RESULTS

Section Properties

In the **Model Builder** window, right-click **Section Properties** and choose **Evaluate>New Table**.

Table 7

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

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

BEAM (BEAM)

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

Cross Section Data 2

1 In the **Physics** toolbar, click **Edges** and choose **Cross Section Data**.

2 Select Edge 1 only.

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. 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 In 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 **General 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 In 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 Right-click **Study 4** and choose **Rename**.

5 In the **Rename Study** dialog box, type `Stationary Study: Beam (Inputs from Beam Cross Section)` in the **New label** text field.

6 Click **OK**.

Step 1: Stationary

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.

1 In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.

2 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

4 From the **Study** list, choose **Stationary Study: Beam Cross Section, Stationary**.

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

6 Click **Add**.

7 In the table, enter the following settings:

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

8 In the **Home** toolbar, click **Compute**.

Compare the von Mises stress for the two cross sections.

Point Evaluation 7

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

2 In the **Settings** window for **Point Evaluation**, type `von Mises 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 **Model>Component 1>Beam>Stress>beam.mises - von Mises stress - N/m²**.

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

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

7 Click **Evaluate**.

8 Locate the **Data** section. From the **Dataset** list, choose **Stationary Study: Beam (Inputs from Beam Cross Section)/Solution 4 (5) (sol4)**.

9 Click **Evaluate**.

Table 8

1 In the **Model Builder** window, under **Results>Tables** click **Table 8**.

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.

STATIONARY STUDY: BEAM

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 Select the **Modify model configuration 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**.

EIGENFREQUENCY STUDY: BEAM

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Eigenfrequency Study: Beam** click **Step 1: Eigenfrequency**.
2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.
3 Select the **Modify model configuration 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 5.5

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 outer ring is 4 mm thick and has an inner radius of 156 mm. The inner ring has an inner radius of 100 mm and a thickness of 11.5 mm.

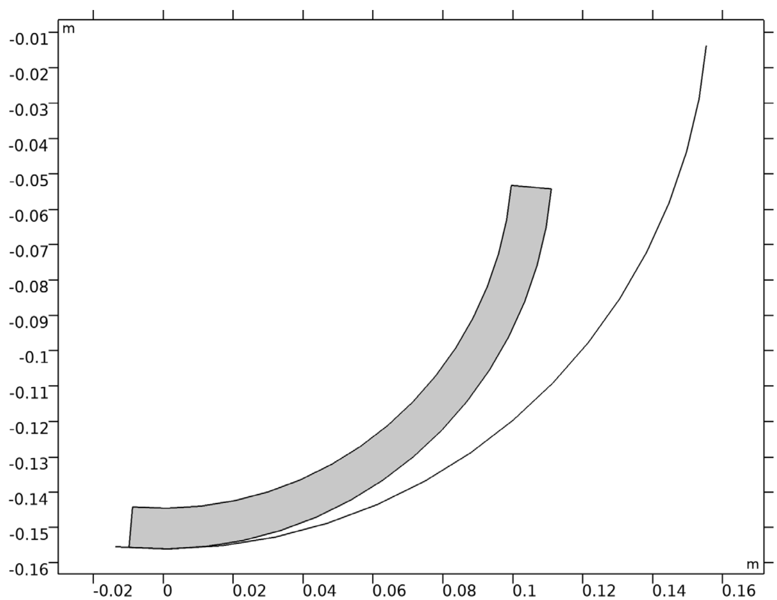


Figure 1: Model geometry.

The outer ring is fixed and rigid. Thus for the contact analysis only its mesh is required, without any physics attached. 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 elevation of the inner ring reaches its maximum value.

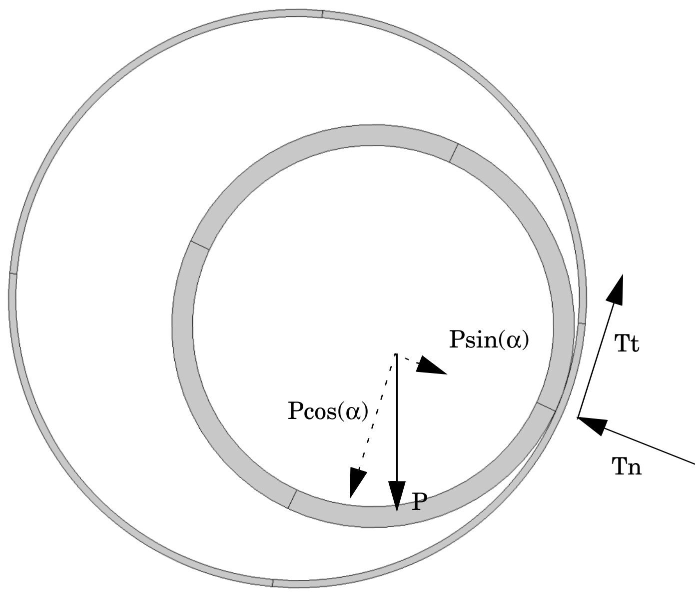


Figure 2: Representation of the contact and friction forces and the resultant of the gravity load.

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.

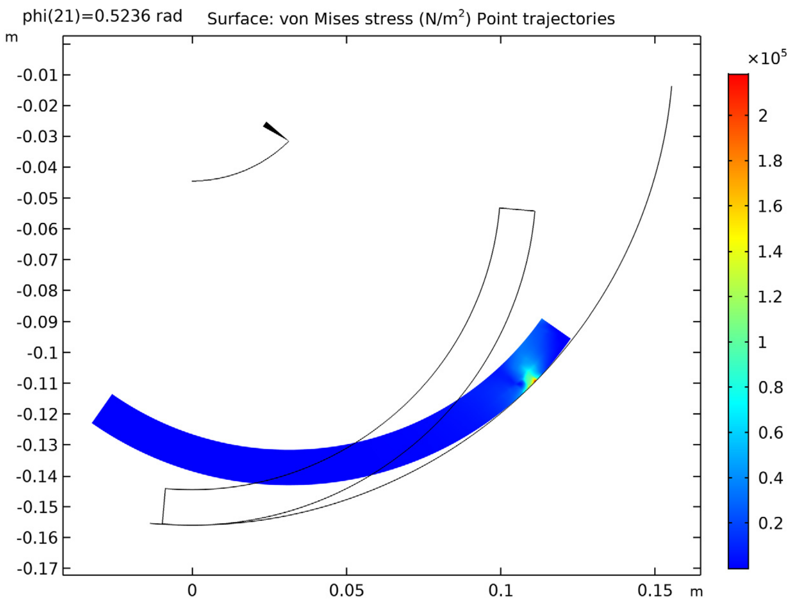


Figure 3: Stress distribution.

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

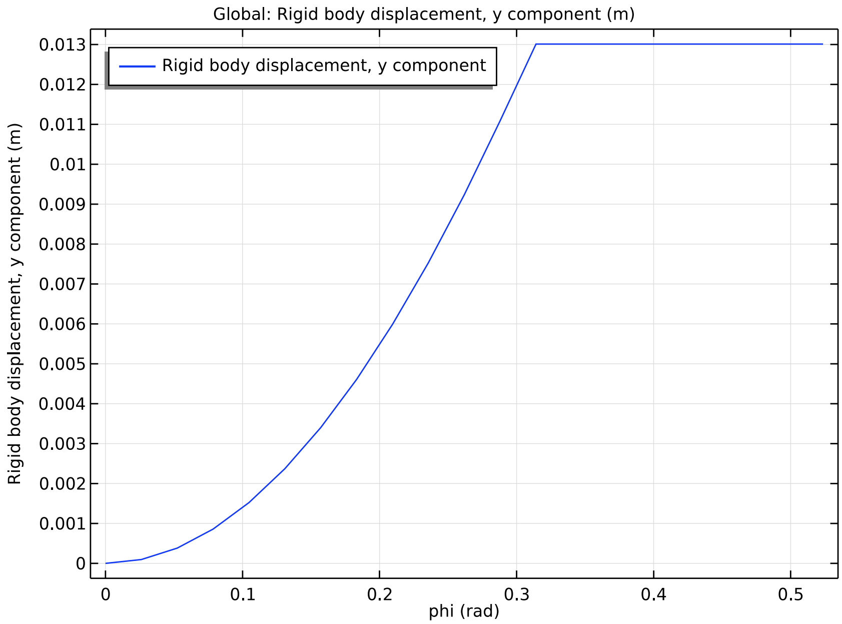


Figure 4: Elevation of the inner ring center versus applied rotation angle.

The computed maximum elevation is about 13 mm, and is in excellent agreement with the analytical solution.

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

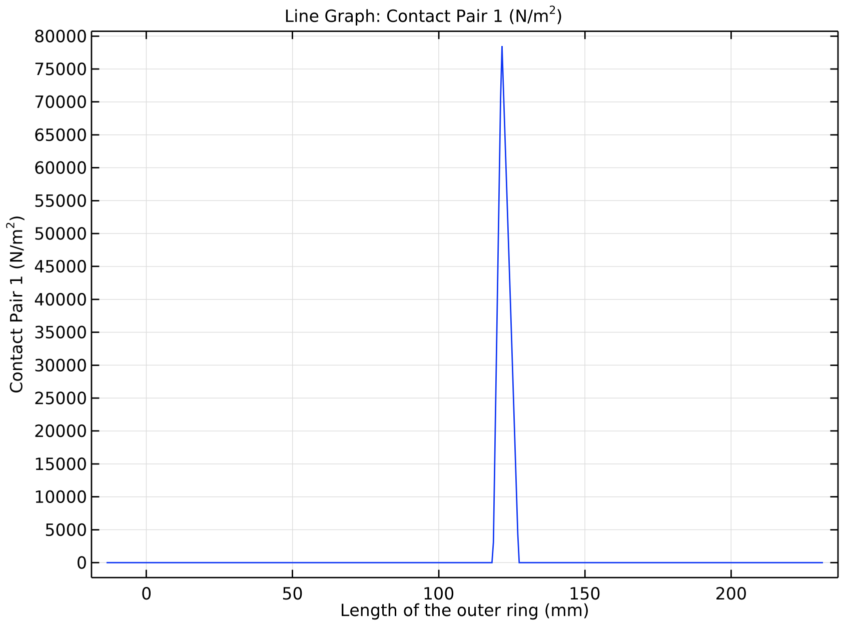


Figure 5: Contact pressure versus length of the outer ring.

Notes About the COMSOL Implementation

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

Since the outer ring is assumed fixed and rigid, it requires no physics. By selecting the *Source external to current physics* check box in the contact node, it is sufficient to define a mesh on its the boundary.

To capture the transition between stick friction and slip friction, a small continuation parameter step is used. Furthermore, the augmented Lagrangian method is better suited for problems dominated by stick-slip friction than the default penalty method, and is thus used in this example.

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 **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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`
`t1`	`4[mm]`	0.004 m	`Outer ring thickness`

Name	Expression	Value	Description
t2	11.5[mm]	0.0115 m	Inner ring thickness
y0	111.5[mm]-156[mm]	-0.0445 m	Inner ring center initial y-position

GEOMETRY 1

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

  Only the inner surface of the outer ring needs to be modeled.

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `r1-t1`.

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

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

7 Click to expand the **Layers** section. Click **Build Selected**.

Circle 2 (c2)

1 In the **Geometry** toolbar, click **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	t2

8 Click **Build Selected**.

Delete Entities 1 (del1)

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

Delete Entities 2 (del2)

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

2 On the object **c1**, select Boundaries 2 and 3 only.

3 In the **Settings** window for **Delete Entities**, click **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 Click **Build Selected**.

DEFINITIONS

Variables 1

1 In 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 1 only.

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

Name	Expression	Unit	Description
L	(r1-t1)*(atan2(-y,-x)-pi/2)	m	Length of the outer ring

Contact Pair 1 (p1)

1 In 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 1 in the **Selection** text field.

5 Click **OK**.

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

7 Select the **Activate selection** toggle button.

8 Click **Paste Selection**.

9 In the **Paste Selection** dialog box, type 4 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	Variable	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)

Contact 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Pairs>Contact**.

Use the **augmented Lagrange** method to evaluate the stick-slip contact.

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

3 From the **Formulation** list, choose **Augmented Lagrangian**.

4 Select the **Source external to current physics** check box.

5 Locate the **Pair Selection** section. Under **Pairs**, click **Add**.

6 In the **Add** dialog box, select **Contact Pair 1 (p1)** in the **Pairs** list.

7 Click **OK**.

Friction 1

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

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

3 In the μ text field, type 1.

4 Click the **Show More Options** button in the **Model Builder** toolbar.

5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

6 Click **OK**.

7 In the **Settings** window for **Friction**, click to expand the **Advanced** section.

8 Select the **Store accumulated slip** check box.

GLOBAL DEFINITIONS

Parameters 1

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

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

2 Select Boundary 5 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 In the **Physics** toolbar, click **Attributes** and choose **Applied Force**.

2 In the **Settings** window for **Applied Force**, locate the **Applied Force** section.

3 Specify the $\mathbf{F}$ vector as

0	x
-500	y

Rigid Connector 1

In the **Model Builder** window, click **Rigid Connector 1**.

Spring Foundation 1

1 In the **Physics** toolbar, click **Attributes** and choose **Spring Foundation**.

2 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

3 In the $\mathbf{k}_u$ text field, type `1e6*(phi==0)`.

4 Locate the **Rotational Spring** section. In the k_θ text field, type `1e6*(phi==0)`.

MESH 1

Mapped 1

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

Distribution 1

1 In the **Model Builder** window, 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 `3`.

Distribution 2

1 In the **Model Builder** window, 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 `60`.

Edge 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Edge**.

2 Select Boundary 1 only.

Distribution 1

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

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

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

4 Click **Build All**.

STUDY 1

Step 1: Stationary

Set up an auxiliary continuation sweep for the `phi` 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 **Results While Solving** section.

3 Select the **Plot** check box.

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

5 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
phi (Inner ring rotation angle)	`range(0,pi/120,pi/6)`	`rad`

Solution 1 (sol1)

1 In 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, 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 **Solid Mechanics**.

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

17 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 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.

Point Trajectories 1

1 In 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

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

RESULTS

1D Plot Group 4

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

2 In the **Settings** window for **1D Plot Group**, locate 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) - m> solid.rig1.v - Rigid body displacement, y component**.

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

1D Plot Group 5

1 In 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 5** and choose **Line Graph**.

2 Select Boundary 1 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 In the **1D Plot Group 5** toolbar, click **Plot**.

1D Plot Group 5

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

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

Edge 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Edge 2D**.

2 Select Boundary 4 only.

Parametric Extrusion 1D 1

In the **Results** toolbar, click **More Datasets** and choose **Parametric Extrusion 1D**.

2D Plot Group 6

1 In the **Results** toolbar, click **2D Plot Group**.

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

3 From the **Dataset** list, choose **Parametric Extrusion 1D 1**.

Surface 1

1 Right-click **2D Plot Group 6** and choose **Surface**.

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>Contact>Friction>solid.sliptot - Accumulated slip - m**.

Height Expression 1

1. Right-click **Surface 1** and choose **Height Expression**.
2. In the **2D Plot Group 6** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.5

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.

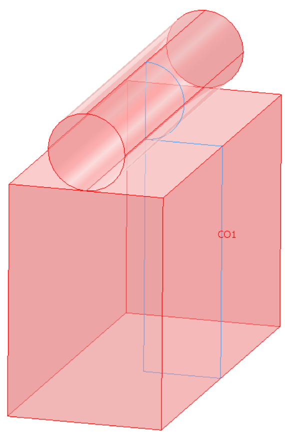


Figure 1: Model geometry.

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 in this example only includes the frictionless part of the example described in Ref. 1. The problem is implemented with the Solid Mechanics interface, and two studies are set up to compare the default penalty contact method and the augmented Lagrangian method.

Figure 2 depicts the deformed shape and the von Mises stress distribution obtained with the penalty contact method.

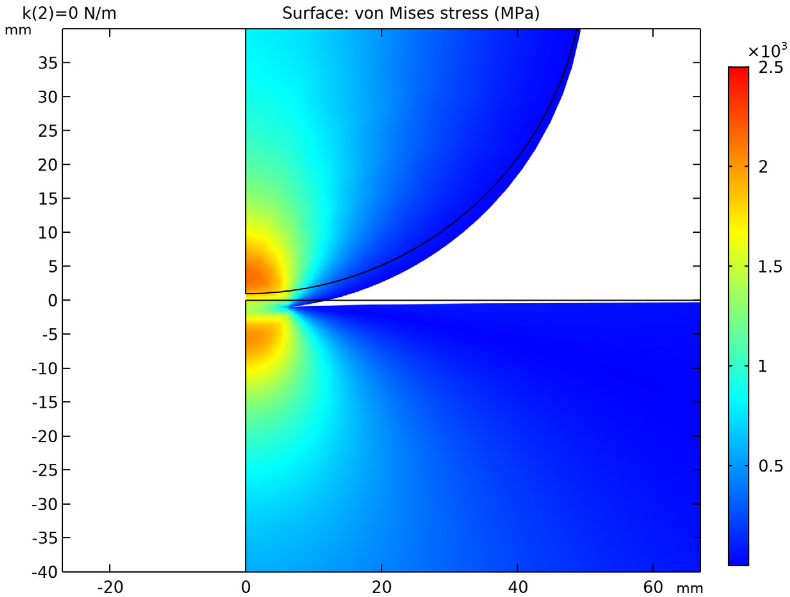


Figure 2: Deformation and von Mises stress at the contact area.

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:

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

$$R' = \lim_{R_2 \to \infty} \frac{R_1R_2}{R_1+R_2} = R_1$$

In these equations, E_1 and E_2 are Young's modulus of the roller and the block, respectively, and R_1 is the radius of the roller. 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 two COMSOL Multiphysics solutions.

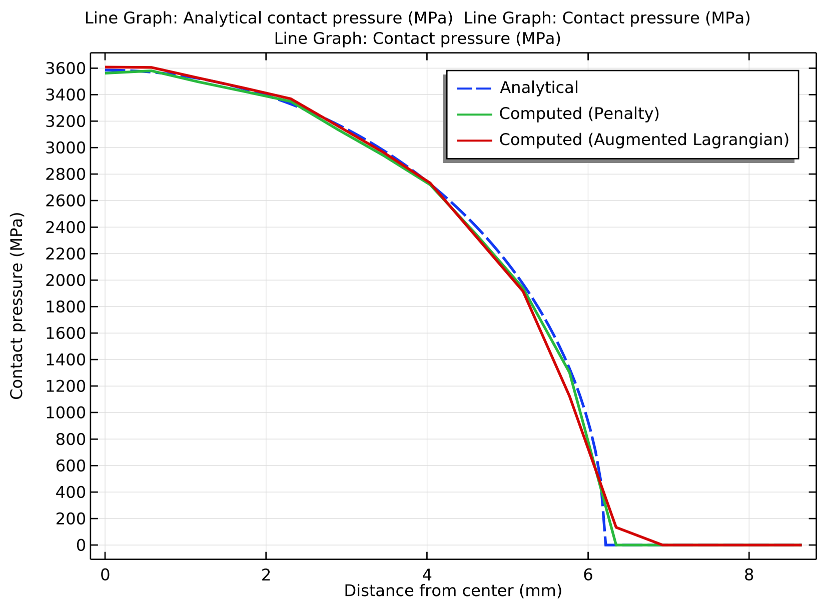


Figure 3: Analytical pressure distribution (dashed line) and COMSOL Multiphysics solution (solid lines).

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.

When using the penalty method, the cylinder is initially stabilized with a weak spring. In a second step, the spring is removed to arrive at the final solution.

To reduce the number of iteration steps and improve convergence when using the augmented Lagrangian method, 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.

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.

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 **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `cylinder_roller_contact.txt`.

DEFINITIONS

Variables 1

1 In 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 In the **Geometry** toolbar, click **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 Click **Build Selected**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **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 Click **Build Selected**.

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

Square 1 (sq1)

1 In the **Geometry** toolbar, click **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 Click **Build Selected**.

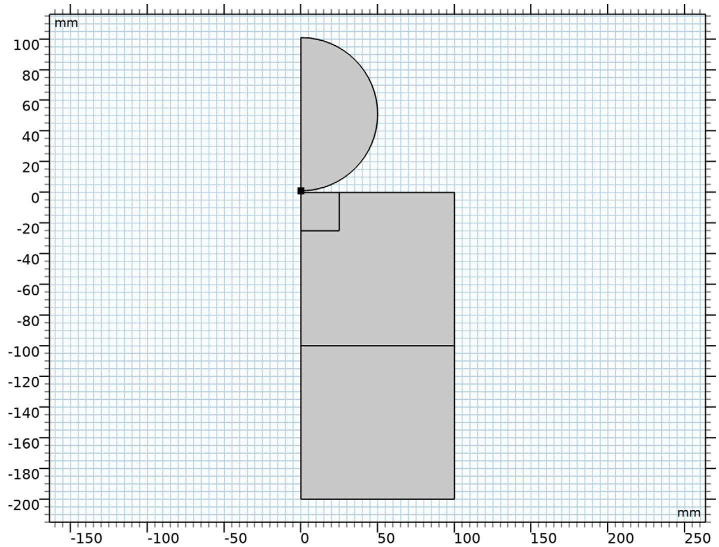
Point 1 (pt1)

1 In the **Geometry** toolbar, click **Point**.

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

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

4 Click **Build Selected**.



Rotate 1 (rot1)

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

2 Select the object **pt1** only.

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

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

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

6 Click **Build Selected**.

Convert to Solid 1 (csol1)

1 In 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 Click **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.

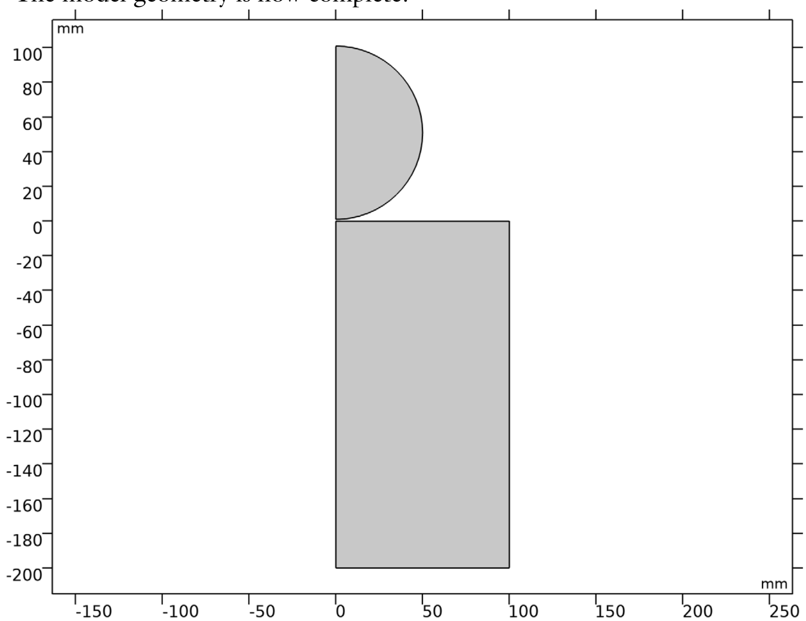
Mesh Control Domains 1 (mcd1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Domains**.

2 On the object **fin**, select Domains 1–3 only.

3 In the **Geometry** toolbar, click **Build All**.

The model geometry is now complete.



DEFINITIONS

Contact Pair 1 (p1)

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

2 Select Boundary 3 only.

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

4 Select the **Activate selection** toggle button.

5 Select Boundary 7 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 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1, 4, and 5 only.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 2 only.

Point Load 1

1 In the **Physics** toolbar, click **Points** and choose **Point Load**.

2 Select Point 5 only.

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

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

4 Specify the $\mathbf{F}_P$ vector as

0	x
-Fn/2	y

Contact 1

1 In 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 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (p1)** in the **Pairs** list.

5 Click **OK**.

Attach a spring to the cylinder in order to prevent rigid body motion before the contact is detected.

Spring Foundation 1

1 In the **Physics** toolbar, click **Points** and choose **Spring Foundation**.

2 Select Point 5 only.

3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

4 In the $\mathbf{k}_P$ text field, type `k`.

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	Variable	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 2 only.

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

4 In the table, enter the following settings:

Property	Variable	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 2 only.

Size 1

1 Right-click **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 7 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**.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, click to expand the **Control Entities** section.

3 Clear the **Smooth across removed control entities** check box.

Distribution 1

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

2 Select Boundaries 3, 10, and 11 only.

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

4 In the **Number of elements** text field, type `20`.

Distribution 2

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

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 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
k (Spring coefficient)	`Fn/dist/5 0`	`N/m`

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

7 In the **Settings** window for **Study**, type `Study 1: Penalty` in the **Label** text field.

8 In the **Home** toolbar, click **Compute**.

RESULTS

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**, locate the **Expression** section.

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

4 In 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 see the stress distribution around the contact region better.

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

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

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

1D Plot Group 4

1 In 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 (k)** list, choose **Last**.

Line Graph 1

1 Right-click **1D Plot Group 4** and choose **Line Graph**.

2 Select Boundary 7 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 **Model>Component 1>Definitions> Variables>p_analytical - Analytical contact pressure - N/m²**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.

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

6 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

7 In the **Width** text field, type `2`.

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

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

10 In the table, enter the following settings:

Legends
`Analytical`

11 In the **1D Plot Group 4** toolbar, click **Plot**.

Line Graph 2

1 Right-click **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 **Model>Component 1>Solid Mechanics> Contact>solid.Tn - Contact pressure - N/m²**.

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

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

Legends
`Computed (Penalty)`

To avoid oscillations in the contact pressure representation, turn off the refinement within the elements.

5 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

1D Plot Group 4

1 In the **Model Builder** window, 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 `Distance from center (mm)`.

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

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

7 In the **1D Plot Group 4** toolbar, click **Plot**.

Now, solve the model using the augmented Lagrangian contact method.

SOLID MECHANICS (SOLID)

Contact 2

1 In 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 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (p1)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Contact**, locate the **Contact Method** section.

7 From the **Formulation** list, choose **Augmented Lagrangian**.

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

ADD STUDY

1 In 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 **General Studies>Stationary**.

4 Click **Add Study**.

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

STUDY 2

Step 1: Stationary

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

2 Select the **Modify model configuration for study step** check box.

3 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 1**.

4 Click **Disable**.

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

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

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

8 In the **Label** text field, type `Study 1: Augmented Lagrangian`.

Solution 2 (sol2)

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

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 1: Augmented Lagrangian> Solver Configurations>Solution 2 (sol2)>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 In the **Study** toolbar, click **Compute**.

The default plot for the second study was disabled. To visualize the stress and contact forces, change the data set in the 2D plot group.

RESULTS

Line Graph 3

1 In the **Model Builder** window, under **Results>1D Plot Group 4** right-click **Line Graph 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 1: Augmented Lagrangian/Solution 2 (sol2)**.

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

Legends
`Computed (Augmented Lagrangian)`

5 In the **1D Plot Group 4** toolbar, click **Plot**.

Prepare the model for later use by disabling the second contact feature in the first study (Penalty).

STUDY 1: PENALTY

Step 1: Stationary

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

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

3 Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 2**.

5 Click **Disable**.

Created in COMSOL Multiphysics 5.5

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.

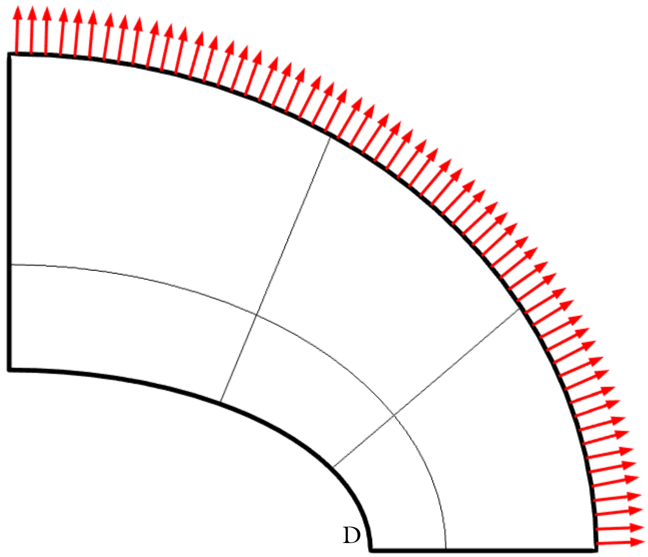


Figure 1: The geometry and load. Only the quarter which is analyzed is shown.

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.)

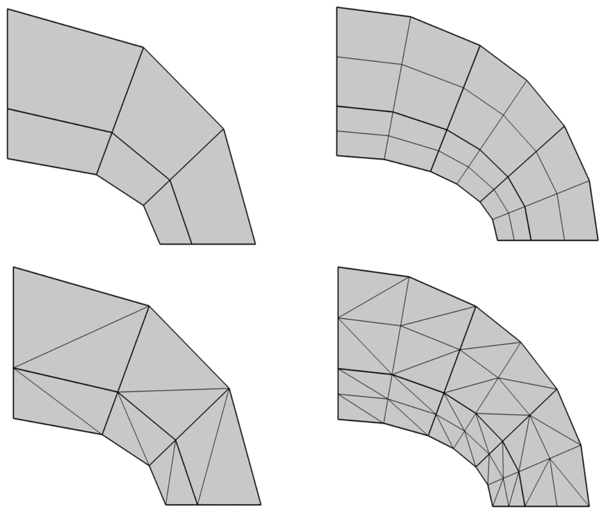


Figure 2: The meshes as specified in Ref. 1. Left column: 'coarse' (div=1). Right column: 'fine' (div=2).

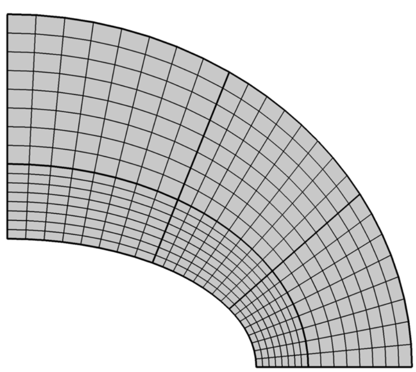


Figure 3: A quadrilateral mesh with div=8.

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

- The interior 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 nontrivial 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.

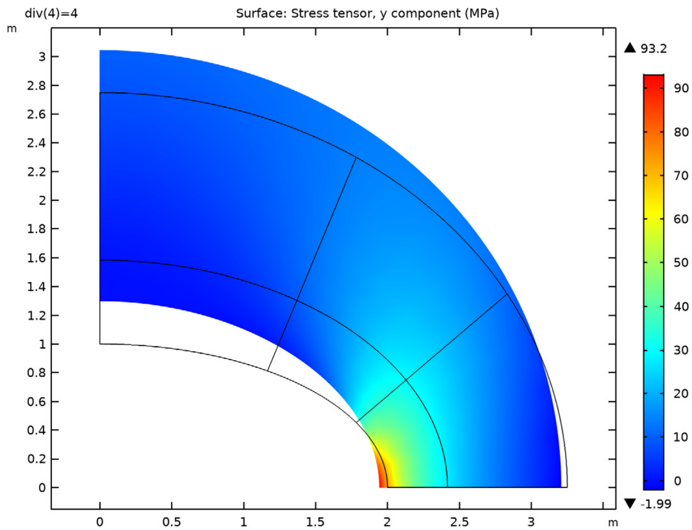


Figure 4: The distribution of the σ_y stress component using div=4 and second order quadrilateral elements.

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 Multiphysics results for the "coarse" and "fine" meshes are given in Table 1.

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

STUDY NUMBER	ELEMENT TYPE	DISCRETIZATION	MESH	COMPUTED VALUE	RELATIVE ERROR
1	Quadrilateral	Linear	Coarse	77.4	-16.5%
1	Quadrilateral	Linear	Fine	88.3	-4.7%
2	Quadrilateral	Quadratic	Coarse	91.9	-0.9%
2	Quadrilateral	Quadratic	Fine	93.4	0.8%
3	Quadrilateral	Cubic	Coarse	94.7	2.2%
3	Quadrilateral	Cubic	Fine	93.0	0.3%
4	Triangle	Linear	Coarse	36.0	-61.1%
4	Triangle	Linear	Fine	55.2	-40.4%
5	Triangle	Quadratic	Coarse	70.7	-23.7%

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

STUDY NUMBER	ELEMENT TYPE	DISCRETIZATION	MESH	COMPUTED VALUE	RELATIVE ERROR
5	Triangle	Quadratic	Fine	85.6	-7.7%
6	Triangle	Cubic	Coarse	81.7	-11.9%
6	Triangle	Cubic	Fine	90.3	-2.6%

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, at least for the quadrilateral elements. With the current mesh, the triangular elements will have a small angle at the stress evaluation point, hence the less accurate result.

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.

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.

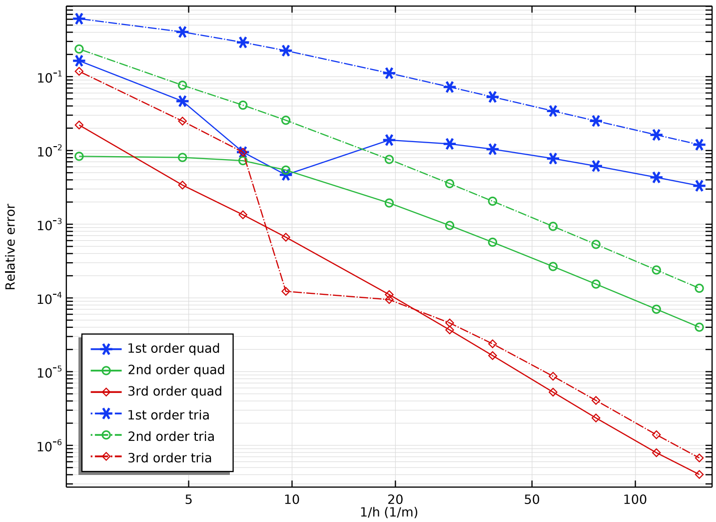


Figure 5: Error with respect to the stress target value as a function of the element size h.

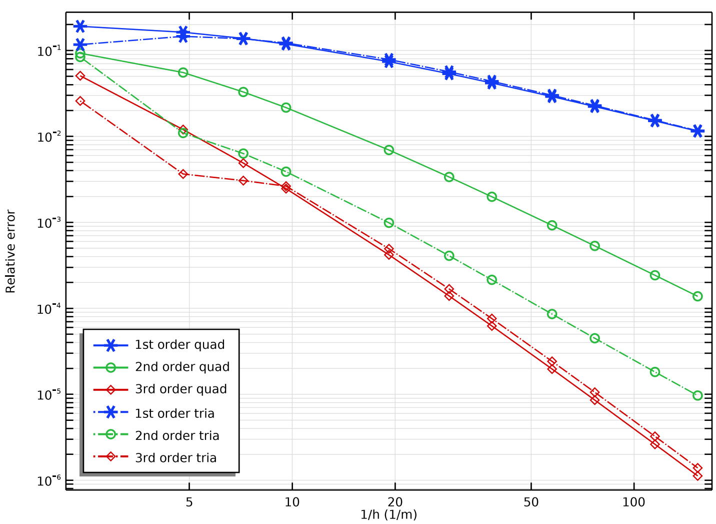


Figure 6: Error in the stress σ_x*. The values are normalized with the target for* σ_y*.*

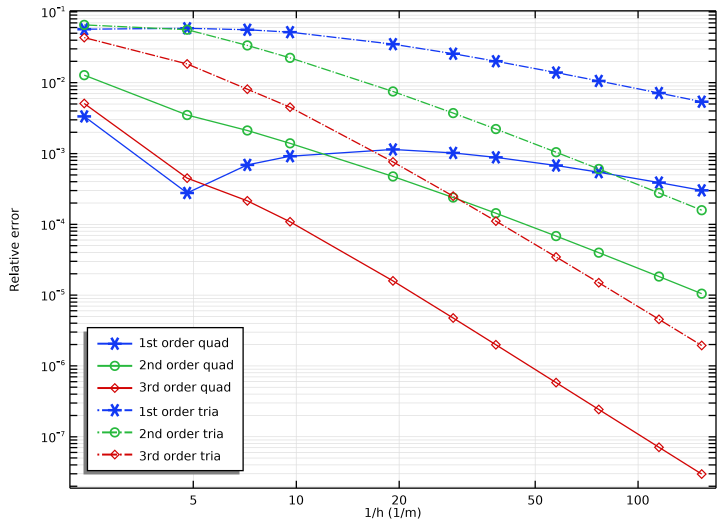


Figure 7: Error in the stress τ_{xy}*. The values are normalized with the target for* σ_y*.*

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.

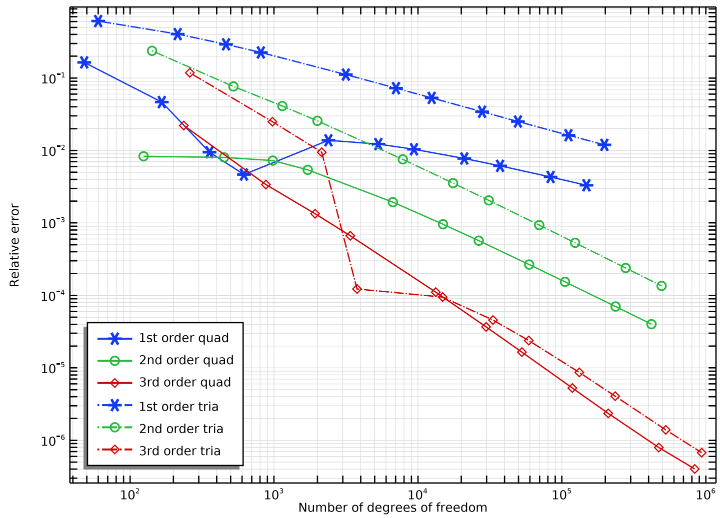


Figure 8: Error with respect to the stress target value as a function of the number of degrees of freedom.

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 **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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]	9.2658E7 Pa	Target stress

GEOMETRY 1

Ellipse 1 (e1)

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

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

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

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

5 Select the object **e3** only.

6 Click **Build All Objects**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

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

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **x** text field, type `1.783` and **y** to `2.3`.

6 Locate the **Endpoint** section. In the **x** text field, type `1.165` and **y** to `0.812`.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

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

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **x** text field, type `2.833` and **y** to `1.348`.

6 Locate the **Endpoint** section. In the **x** text field, type `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	Variable	Value	Unit	Property group
Young's modulus	E	210E3[MPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	0	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 `0.1`.

Symmetry 1

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

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

Boundary Load 1

1 In 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

Mapped 1

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

Distribution 1

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

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

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

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

5 Click **Build All**.

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

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**, click to expand the **Discretization** section.

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

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

4 Click the **Show More Options** button in the **Model Builder** toolbar.

5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

6 Click **OK**.

Discretization 1

1 In 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 **Label** text field, type `Discretization Linear`.

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 **Label** text field, type `Discretization Cubic`.

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

1 In 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
div (Mesh refinement factor)	1 2 3 4 8 12 16 24 32 48 64	

Step 1: Stationary

1 In the **Model Builder** window, 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 (solid)	√	Discretization Linear

ROOT

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

ADD STUDY

1 In 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 **General 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 **General 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**.

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 (solid)	√	Discretization Cubic

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

Convert 1

Right-click **Component 1 (comp1)>Meshes>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 **General 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.

Parametric Sweep

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

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 (solid)	√	Discretization Linear

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **General 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.

Parametric Sweep

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 **General Studies>Stationary**.

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

4 In 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**.

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 (solid)	√	Discretization Cubic

STUDY QUAD LINEAR

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

RESULTS

1D Plot Group 3

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

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

Point Graph 1

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

2 Select Point 11 only.

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

4 From the **Dataset** list, choose **Study Quad Linear/Parametric Solutions 1 (sol2)**.

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

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

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

8 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

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

10 Click to expand 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
`1st order quad`

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

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

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

Legends
`2nd order quad`

Point Graph 3

1 Right-click **Point Graph 2** and choose **Duplicate**.

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

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

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

Legends
`3rd order quad`

Point Graph 4

1 Right-click **Point Graph 3** and choose **Duplicate**.

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

3 From the **Dataset** 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 Right-click **Point Graph 4** and choose **Duplicate**.

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

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

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

Legends
`2nd order tria`

Point Graph 6

1 Right-click **Point Graph 5** and choose **Duplicate**.

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

3 From the **Dataset** 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`

Point Graph 4

1 In the **Model Builder** window, click **Point Graph 4**.

2 In the **Settings** window for **Point Graph**, locate the **Coloring and Style** section.

3 From the **Color** list, choose **Cycle (reset)**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle (reset)**.

Mesh convergence sy at D

1 In the **Model Builder** window, click **Mesh convergence sy at D**.

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 Locate the **Axis** section. Select the **x-axis log scale** check box.

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

10 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

11 In 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 In the **Mesh convergence sx at D** toolbar, click **Plot**.

Mesh convergence sx at D 1

1 In the **Model Builder** window, 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 In the **Mesh convergence sxy at D** toolbar, click **Plot**.

Mesh convergence sy at D 1

1 In the **Model Builder** window, 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`.

4 Do the same modification for all graphs from **Point Graph 2** to **Point Graph 6** according to the following table:

5 In the table, enter the following settings:

Name	x-Axis Data Expression
Point Graph 2	`12*div^2*4+10*div*2+2+6*div^2*9`
Point Graph 3	`12*div^2*9+10*div*3+2+6*div^2*16`
Point Graph 4	`12*div^2*1+10*div*1+2+6*div^2*2*3`
Point Graph 5	`12*div^2*4+10*div*2+2+6*div^2*2*6`
Point Graph 6	`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, 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 In the **Mesh convergence sy at D (by DOFs)** toolbar, click **Plot**.

Point Evaluation 1

1 In 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 **Dataset** 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) - N/m²> 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, quad linear

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 **Dataset** list, choose **Study Quad Quadratic/Parametric Solutions 2 (sol15)**.

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

Expression	Unit	Description
solid.sy	MPa	Stress, quad quadratic

5 Click **Evaluate**.

Point Evaluation 3

1 Right-click **Point Evaluation 2** and choose **Duplicate**.

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

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

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

Expression	Unit	Description
solid.sy	MPa	Stress, quad cubic

5 Click **Evaluate**.

Point Evaluation 4

1 Right-click **Point Evaluation 3** and choose **Duplicate**.

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

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

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

Expression	Unit	Description
solid.sy	MPa	Stress, tria linear

5 Click **Evaluate**.

Point Evaluation 5

1 Right-click **Point Evaluation 4** and choose **Duplicate**.

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

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

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

Expression	Unit	Description
solid.sy	MPa	Stress, tria quadratic

5 Click **Evaluate**.

Point Evaluation 6

1 Right-click **Point Evaluation 5** and choose **Duplicate**.

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

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

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

Expression	Unit	Description
solid.sy	MPa	Stress, tria cubic

5 Click **Evaluate**.

Stress (solid)

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

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

3 From the **Dataset** 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) - N/m²>solid.sy - Stress tensor, y component**.

3 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

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

Stress (solid)

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

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

3 Select the **Show maximum and minimum values** check box.

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

Created in COMSOL Multiphysics 5.5

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 an ordinary Linear Elastic Material model in the Shell interfaces available with the Structural Mechanics Module. The same example can be modeled using a Layered Linear Elastic Material model in the Shell interface. The model using the latter approach can be found in the Verification Examples folder of the Composite Materials Application Library.

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.

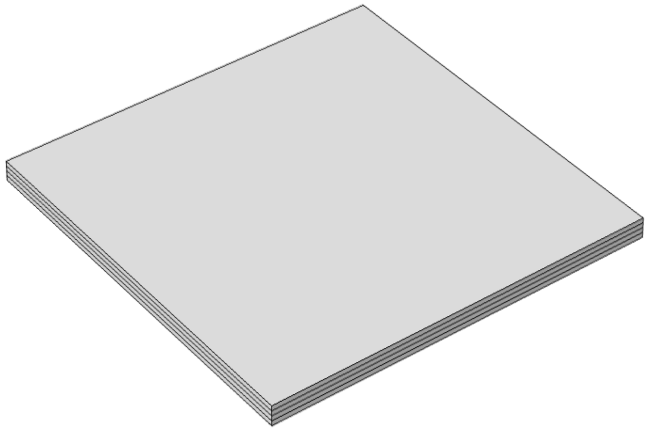


Figure 1: Geometry of layered shell with ply orientations 90/-45/45/0 from top to bottom.

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	u, v, w, 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	u, 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 nonzero 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 nonzero 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 either 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

$$\mathrm{FI} = \sigma_i F_{ij} \sigma_j + \sigma_i f_i \tag{3}$$

where σ_i is the 6-by-1 stress vector (sorted using Voigt notation), F_{ij} is a 6-by-6 symmetric matrix (fourth rank tensor) that contains the coefficients for the quadratic terms, and f_i is a 6-by-1 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

$$a\,\mathrm{SF}^2 + b\,\mathrm{SF} = 1 \tag{4}$$

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. 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 nonzero 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 nonzero elements of the matrix F_{ij} are given by the Equation 5.

$$
\begin{aligned}
&F_{ii} = \frac{1}{\sigma_{ti}^2}; \quad i = 1, 2, 3 \\
&F_{44} = \frac{1}{\sigma_{ss23}^2}, \quad F_{55} = \frac{1}{\sigma_{ss13}^2}, \quad F_{66} = \frac{1}{\sigma_{ss12}^2} \\
&F_{ij} = -\frac{1}{2}(F_{ii} + F_{jj} - F_{kk}); \quad i \neq j \neq k, i = 1, 2, 3
\end{aligned}
\tag{5}
$$

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 nonzero 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 nonzero 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 midplane 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 through 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 midplane 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.

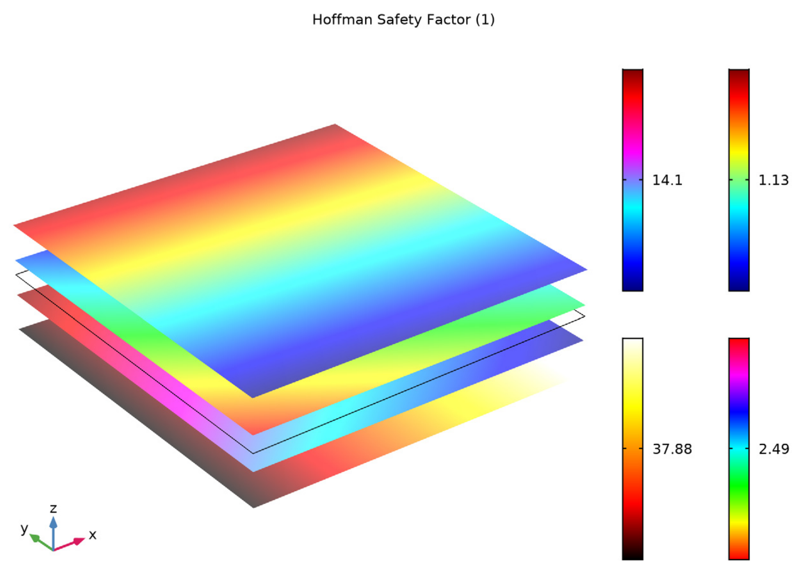


Figure 2: Hoffman safety factors at midplanes for a stack of shells.

The von Mises stresses in all plies are shown in Figure 3. The stress in ply 1 is the lowest, but this layer is still more susceptible to failure due to the orientation of its fibers.

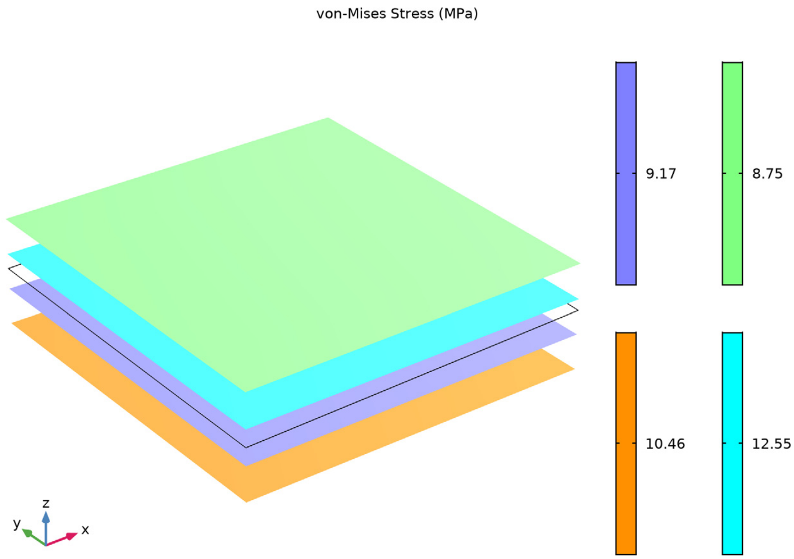


Figure 3: von Mises stress in a stack of shells.

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 midsurface 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_Mechanics_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 **General Studies>Stationary**.

12 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Load the text file containing the material properties and material strengths.

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `failure_prediction_in_a_layered_shell_materialproperties.txt`.

DEFINITIONS

Set up three rotated coordinate systems.

Rotated System 2 (sys2)

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Rotated System**.

2 In the **Settings** window for **Rotated System**, locate the **Rotation** 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 **Rotation** 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 **Rotated System 3 (sys3)** and choose **Duplicate**.

2 In the **Settings** window for **Rotated System**, locate the **Rotation** section.

3 Find the **Euler angles (Z-X-Z)** subsection. In the α text field, type `pi/4`.

GEOMETRY 1

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Square 1 (sq1)

1 In the **Work Plane** toolbar, click **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 in the **Graphics** toolbar.

MATERIALS

Material 1 (mat1)

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

SHELL (SHELL)

Activate **Advanced Physics** option from **Show** button.

1. Click the **Show More Options** button in the **Model Builder** toolbar.
2. In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.
3. Click **OK**.

   The layered shell is modeled using four separate shell interfaces located on the same boundary (mesh surface), sharing the degrees of freedom. The stacking of the shells is done using a **Physical Offset** option. With this option the constraints and loads are transferred 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 the 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 the midplane of each shell layer. Set the **Default Through-Thickness Result Location** to zero for all shells.
4. In the **Settings** window for **Shell**, type `Ply 1` in the **Label** text field.
5. In the **Name** text field, type `shell1`.
6. Click to expand the **Default Through-Thickness Result Location** section. In the z text field, type `0`.
7. Click to expand the **Discretization** section. From the **Displacement field** list, choose **Linear**.

Thickness and Offset 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Ply 1 (shell1)** click **Thickness and Offset 1**.
2. In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.
3. 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`.

Linear Elastic Material 1

Choose the orthotropic solid model for the linear elastic material and assign **Rotated System 2** as **Shell Local System**.

1 In the **Model Builder** window, 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)

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 **Discretization** section. From the **Displacement field** list, choose **Linear**.

4 Locate the **Default Through-Thickness Result Location** section. In the z text field, type `0`.

5 Click to expand the **Dependent Variables** section. In the **Displacement field** text field, type `u`.

6 In the **Displacement of shell normals** text field, type `ar`.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Ply 2 (shell2)** click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

3 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`.

Linear Elastic Material 1

Choose the orthotropic solid model for the linear elastic material and assign **Rotated System 3** as **Shell Local System**.

1 In the **Model Builder** window, 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)

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 **Discretization** section. From the **Displacement field** list, choose **Linear**.

4 Locate the **Default Through-Thickness Result Location** section. In the z text field, type `0`.

5 Locate the **Dependent Variables** section. In the **Displacement field** text field, type `u`.

6 In the **Displacement of shell normals** text field, type `ar`.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Ply 3 (shell3)** click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

3 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`.

Linear Elastic Material 1

Choose the orthotropic solid model for the linear elastic material and assign **Rotated System 4** as **Shell Local System**.

1 In the **Model Builder** window, 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)

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 **Discretization** section. From the **Displacement field** list, choose **Linear**.

4 Locate the **Default Through-Thickness Result Location** section. In the z text field, type `0`.

5 Locate the **Dependent Variables** section. In the **Displacement field** text field, type `u`.

6 In the **Displacement of shell normals** text field, type `ar`.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Ply 4 (shell4)** click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

3 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`.

Linear Elastic Material 1

1 In the **Model Builder** window, 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 the 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	Variable	Value	Unit	Property group
Young's modulus	{Evector1, Evector2, Evector3}	`{E1,E2, E3}`	Pa	Orthotropic
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	`{nu12, nu23, nu13}`	1	Orthotropic
Shear modulus	{Gvector1, Gvector2, Gvector3}	`{G,G,G}`	N/m^2	Orthotropic
Density	rho	`7800`	kg/m^3	Basic

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 In the **Physics** toolbar, click **Attributes** and choose **Safety**.

2 In the **Settings** window for **Safety**, type `Safety: Tsai-Wu Orthotropic Criterion` in the **Label** text field.

3 Locate the **Failure Model** section. 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 **Safety 1** 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 the 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	Variable	Value	Unit	Property group
Tensile strengths	{sigmats1, sigmats2, sigmats3}	`{Sigmats1, Sigmats2, Sigmats3}`	Pa	Orthotropic strength parameters, Voigt notation
Compressive strengths	{sigmacs1, sigmacs2, sigmacs3}	`{Sigmacs1, Sigmacs2, Sigmacs3}`	Pa	Orthotropic strength parameters, Voigt notation
Shear strengths	{sigmass1, sigmass2, sigmass3}	`{Sigmass23, Sigmass13, Sigmass12}`	Pa	Orthotropic strength parameters, Voigt notation

Property	Variable	Value	Unit	Property group
Second rank tensor, Voigt notation	{F_s1, F_s2, F_s3, F_s4, F_s5, F_s6}	`{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_f11, F_f12, F_f22, F_f13, F_f23, F_f33, F_f14, F_f24, F_f34, F_f44, F_f15, F_f25, F_f35, F_f45, F_f55, F_f16, F_f26, F_f36, F_f46, F_f56, F_f66} ; F_fij = F_fji	`{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	{Evector1, Evector2, Evector3}	`{207e9,7.6e9,7.6e9}`	Pa	Orthotropic
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	`{0.3,0,0}`	1	Orthotropic
Shear modulus	{Gvector1, Gvector2, Gvector3}	`{5e9,5e9,5e9}`	N/m^2	Orthotropic

Property	Variable	Value	Unit	Property group
Loss factor for orthotropic Young's modulus	{eta_Evector1, eta_Evector2, eta_Evector3}	{0,0,0}	1	Orthotropic
Loss factor for orthotropic shear modulus	{eta_Gvector1, eta_Gvector2, eta_Gvector3}	{0,0,0}	1	Orthotropic

PLY 1 (SHELL1)

Fixed Constraint 1

1 In 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 In 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 a fixed x-direction translation on Node 2, apply the displacement `u0` in the x direction to Point 2 of `shell2`, and the displacement `-u0` in the x direction to the same point of shell3. Also add a **Global Equation** node under shell3 for the additional degree of freedom u0.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Ply 2 (shell2)**.

Prescribed Displacement/Rotation 1

1 In 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_{0x} text field, type u0.

PLY 3 (SHELL3)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Ply 3 (shell3)**.
2 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.

Prescribed Displacement/Rotation 1

1 Select Point 2 only.
2 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.
3 Select the **Prescribed in x direction** check box.
4 In the u_{0x} text field, type -u0.
5 Click the **Show More Options** button in the **Model Builder** toolbar.
6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.
7 Click **OK**.

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **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. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `displacement` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Displacement (m)**.

8 Click **OK**.

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 Select Boundary 1 only.

Distribution 1

1 Right-click **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 **Model Builder** window, click **Study 1**.

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

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

4 In the **Home** toolbar, click **Compute**.

RESULTS

In the **Model Builder** window, expand the **Results** node.

Cut Point 3D 1

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

2 In the **Settings** window for **Point Evaluation**, type `Failure indices in Ply 1` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.

4 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	

5 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 `Failure indices in Ply 1` in the **Label** text field.

Point Evaluation 2, 3, 4

Create three similar **Point Evaluation** nodes by duplicating the **Point Evaluation 1** 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 point evaluation nodes and tables appropriately.

Point Evaluation 5

1 In 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 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.

4 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	

5 Click **Evaluate**.

Table 5

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

2 In the **Settings** window for **Table**, type `Safety factors in Ply 1` in the **Label** text field.

Point Evaluation 6, 7, 8

Create three similar **Point Evaluation** nodes by duplicating the **Point Evaluation 5** 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 In 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 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.

4 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`	

5 Click **Evaluate**.

Table 9

1 In the **Model Builder** window, under **Results>Tables** click **Table 9**.

2 In the **Settings** window for **Table**, type `Stresses in Ply 1` in the **Label** text field.

Point Evaluation 10, 11, 12

Create three similar **Point Evaluation** nodes by duplicating the **Point Evaluation 9** 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 In 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.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `von-Mises Stress (MPa)`.

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 `round(shell1.mises)`.

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

Deformation 1

1 Right-click **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 **Surface 1** 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, 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 in the **Graphics** toolbar.

To visualize the Hoffman safety factors in the layered shell, use four different **Surface** plots for the 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 In 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.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Hoffman Safety Factor (1)`.

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 **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, 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**.

Created in COMSOL Multiphysics 5.5

Eigenfrequency Analysis of a Free Cylinder

Introduction

In the following example you compute the eigenfrequencies of a free circular pipe using three different approaches:

- An axisymmetric model using the Solid Mechanics interface.
- An axisymmetric model using the Shell interface.
- A sector of a 3D model using cyclic symmetry in the Solid Mechanics interface.

The example is taken from NAFEMS *Free Vibration Benchmarks* (Ref. 1). The eigenfrequencies are compared with the values given in the benchmark report.

As an extension, you will also compute eigenfrequencies with twisting deformation.

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.

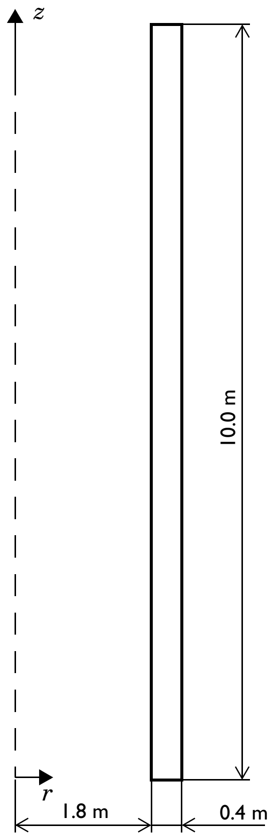


Figure 1: Model geometry in the rz-plane.

In the axisymmetric solid model, the geometry consists of this rectangle.

In the axisymmetric shell interface, the mesh is placed on the line representing the inner boundary of the cylinder, and an offset property is used in order to account for the fact that the shell model should represent the midsurface.

In the 3D solid model, the rectangle is swept around the axis of revolution, so that a 15° sector is formed. As long as 360° is as an exact multiple of the sector angle, any angle could have been used.

MATERIAL

The material is isotropic linear elastic with $E = 2.0 \cdot 10^{11}$ Pa, $\nu = 0.3$, and $\rho = 8000\ \mathrm{kg/m^3}$.

LOADS

In an eigenfrequency analysis loads are not needed.

CONSTRAINTS

In the axisymmetric models, no constraints are applied because the cylinder is free. In the 3D solid model, cyclic symmetry constraints are applied to the cuts in the azimuthal direction.

Results

For structural mechanics, there are two possible interpretations of axisymmetry. The most common one is that there are no displacements out of the RZ-plane. Another interpretation, which also allows twisting motion, is that all derivatives of the displacements with respect to the azimuthal coordinate is zero. Such an extension is available when using the Solid Mechanics interface.

The original NAFEMS example does not contain out-of-plane displacements, in which case there is one rigid body mode. The rigid body mode with an eigenvalue close to zero is found in all physics interfaces. 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	SOLID MECHANICS, AXISYMMETRY	SHELL , AXISYMMETRY	SOLID MECHANICS, 3D	TARGET (Ref. 1)
f_2	243.50	243.64	243.50	243.53
f_3	377.39	378.16	377.39	377.41
f_4	394.21	394.11	394.22	394.11
f_5	397.84	397.36	397.84	397.72
f_6	405.36	407.43	405.36	405.28

The analytical solution for twisting vibration of a free cylindrical pipe is

$$f_n = \frac{n}{2L}\sqrt{\frac{G}{\rho}} \tag{1}$$

Here, G is the shear modulus,

$$G = \frac{E}{2(1+\nu)} \tag{2}$$

In this case, there is one more rigid body mode: pure rotation around the axis of revolution. The computed non-trivial eigenfrequencies have a very good agreement with the analytical solution:

EIGENFREQUENCY	SOLID MECHANICS, AXISYMMETRY	SOLID MECHANICS, 3D	TARGET (ANALYTICAL)
f_1	155.04	155.04	155.04
f_2	310.09	310.09	310.09

Figure 2 shows the shape of the second eigenmode in the axisymmetric solid model. In Figure 3, the same plot is shown for the axisymmetric shell interface. In both cases, **Revolution 2D** data sets have been used for extending the axisymmetric model into 3D space..

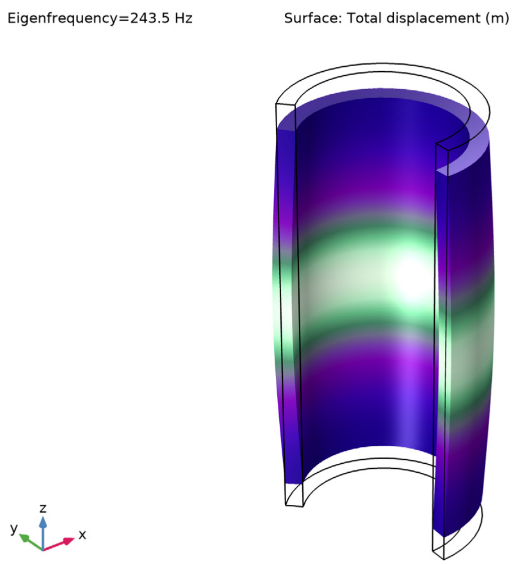


Figure 2: The second non-rigid eigenmode, computed using an axisymmetric solid mechanics interface.

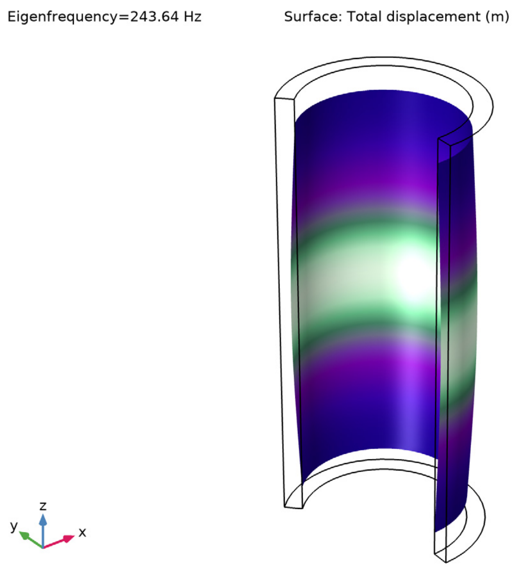


Figure 3: The first non-rigid eigenmode, computed using an axisymmetric shell interface. Due to the offset property, the shell is modeled at the true midsurface, even though the mesh is at the inner boundary of the cylinder.

In Figure 4 and Figure 5, two eigenmodes from the 3D solid model are shown. A **Sector 3D** data set has been used for expanding the results from the original 15° sector.

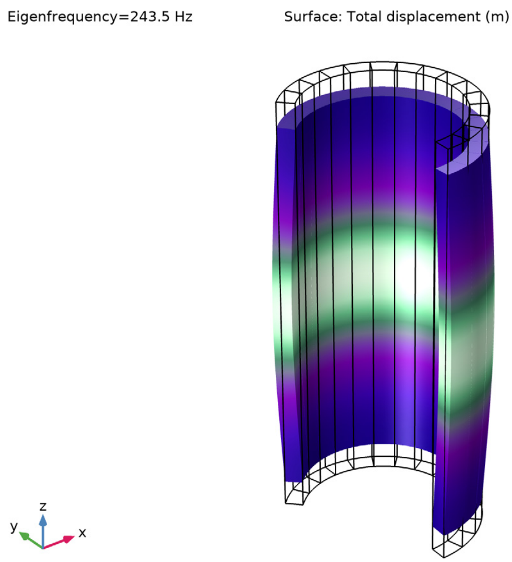


Figure 4: The second non-rigid eigenmode, computed using a 3D solid mechanics interface with cyclic symmetry boundary conditions.

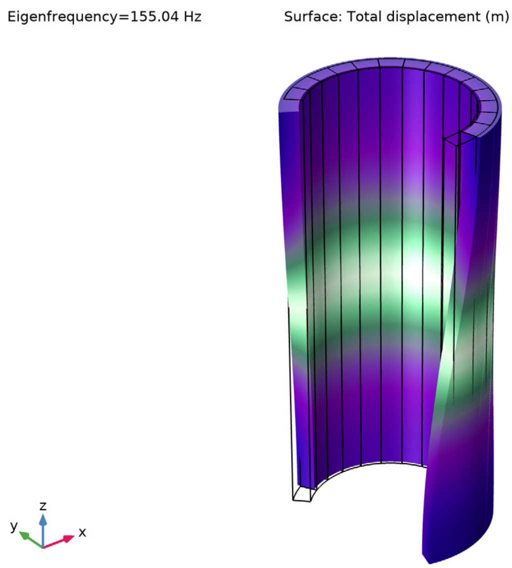


Figure 5: The first non-rigid eigenmode, computed using a 3D solid mechanics interface with cyclic symmetry boundary conditions.

Notes About the COMSOL Implementation

In the 3D solid model, you could have used ordinary **Symmetry** boundary conditions instead of the **Periodic Condition**. The effect would have been that only the in-plane modes were computed.

In a real pipe, there are however also other eigenmodes, which are not axially symmetric. You can find such modes by using azimuthal mode numbers other than zero in the settings for the cyclic symmetry condition (3D) and Solid Mechanics interface settings (2D axisymmetry). Such modes can be visualized by setting the azimuthal mode number to the corresponding value in the **Advanced** section in the settings for the **Revolution 2D** and **Sector 3D** data sets.

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 **General Studies>Eigenfrequency**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
height	10[m]	10 m	Height of cylinder
thic	0.4[m]	0.4 m	Thickness of cylinder
r_in	1.8[m]	1.8 m	Inner radius

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

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

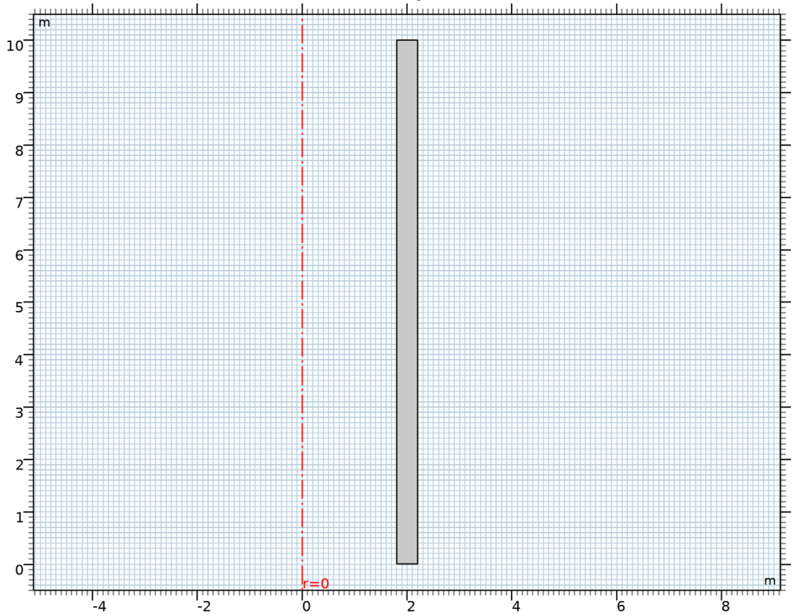
3 In the **Width** text field, type `thic`.

4 In the **Height** text field, type `height`.

5 Locate the **Position** section. In the **r** text field, type `r_in`.

6 Click **Build All Objects**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar.



GLOBAL DEFINITIONS

In this example, the same material data will be referenced from several physics interfaces, so it is convenient to define a global material.

Material 1 (mat1)

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, click to expand the **Material Properties** section.

3 In the **Material properties** tree, select **Basic Properties>Density**.

4 Click **Add to Material**.

5 In the **Material properties** tree, select **Solid Mechanics>Linear Elastic Material>Young's Modulus and Poisson's Ratio**.

6 Click **Add to Material**.

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

Property	Variable	Value	Unit	Property group
Density	rho	8000	kg/m³	Basic
Young's modulus	E	2e11	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio

MATERIALS

Material Link 1 (matlnk1)

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

MESH 1

Mapped 1

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

Distribution 1

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `20`.

4 Select Boundary 1 only.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `2`.

4 Select Boundary 2 only.

5 Click **Build All**.

STUDY 1

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

2 In the **Settings** window for **Study**, type `Study 1, 2D axisymmetric solid` in the **Label** text field.

3 In the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (solid)

Visualize an eigenmode in 3D.

Mode Shape, 3D (solid)

1 In the **Model Builder** window, 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 Click the **Show Grid** button in the **Graphics** toolbar.

5 In the **Mode Shape, 3D (solid)** toolbar, click **Plot**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

COMPONENT 1 (COMP1)

Add a Shell interface with the same data, and compute the eigenfrequencies.

ADD PHYSICS

1 In 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>Shell (shell)**.

4 Click **Add to Component 1** in the window toolbar.

5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

SHELL (SHELL)

Select Boundary 1 only.

Thickness and Offset 1

Since the inner boundary of the cylinder is used as geometry for the shell interface, you must use an offset to position the midsurface at the correct radial coordinate.

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

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

4 From the **Offset definition** list, choose **Relative offset**.

5 In the $z_{\mathrm{reloffset}}$ text field, type `-1`.

MATERIALS

Material Link 2 (matlnk2)

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

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

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

4 Select Boundary 1 only.

ADD STUDY

1 In 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 **General Studies>Eigenfrequency**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Solid Mechanics** interface.

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

6 In 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**, type `Study 2, 2D axisymmetric shell` in the **Label** text field.

3 In the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape, 3D (shell)

1 In the **Model Builder** window, under **Results** click **Mode Shape, 3D (shell)**.

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

3 From the **Eigenfrequency (Hz)** list, choose **243.64**.

4 Click the **Show Grid** button in the **Graphics** toolbar.

5 In the **Mode Shape, 3D (shell)** toolbar, click **Plot**.

ROOT

Now, add a 3D solid sector with cyclic symmetry boundary conditions and compute the eigenfrequencies.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

GEOMETRY 2

Work Plane 1 (wp1)

1. In 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**.

GEOMETRY 1

Rectangle 1 (r1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Rectangle 1 (r1)** and choose **Copy**.

GEOMETRY 2

Plane Geometry

In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2> Work Plane 1 (wp1)** click **Plane Geometry**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

Right-click **Plane Geometry** and choose **Paste Rectangle**.

Revolve 1 (rev1)

1. In 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 15.
5. Click **Build All Objects**.

ADD PHYSICS

1. In 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>Solid Mechanics (solid)**.
4 Click **Add to Component 2** in the window toolbar.
5 In the **Model Builder** window, click **Component 2 (comp2)**.
6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

SOLID MECHANICS 2 (SOLID2)

Periodic Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.
2 Select Boundaries 2 and 5 only.
3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.
4 From the **Type of periodicity** list, choose **Cyclic symmetry**.

MESH 2

Mapped 1

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Mesh 2** and choose **More Operations>Mapped**.
2 Select Boundary 3 only.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type 2.
4 Select Edges 2 and 7 only.

Mapped 1

1 In the **Model Builder** window, click **Mapped 1**.
2 Click **Build Selected**.

Swept 1

In the **Model Builder** window, right-click **Mesh 2** and choose **Swept**.

Distribution 1

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type 20.
4 Click **Build All**.

MATERIALS

Material Link 3 (matlnk3)

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **More>Material Link**.

ADD STUDY

1 In 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 **General Studies> Eigenfrequency**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Solid Mechanics** and **Shell** interfaces.

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

6 From the **Home** menu, choose **Add Study**.

STUDY 3

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

2 In the **Settings** window for **Study**, type `Study 3, 3D solid sector` in the **Label** text field.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 3, 3D solid sector** 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 `10`.

5 In the **Model Builder** window, collapse the **Study 3, 3D solid sector** node.

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

RESULTS

Mode Shape (solid2)

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

2 From the **Eigenfrequency (Hz)** list, choose **243.5**.

3 In the **Mode Shape (solid2)** toolbar, click **Plot**.

Datasets

In the **Model Builder** window, click **Datasets**.

Sector 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Sector 3D**.
2 In the **Settings** window for **Sector 3D**, locate the **Symmetry** section.
3 In the **Number of sectors** text field, type 360/15.
4 From the **Sectors to include** list, choose **Manual**.
5 In the **Start sector** text field, type 18.
6 In the **Number of sectors to include** text field, type 15.

Mode Shape (solid2)

1 In the **Model Builder** window, click **Mode Shape (solid2)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Dataset** list, choose **Sector 3D 1**.
4 Click the **Zoom Extents** button in the **Graphics** toolbar.
5 In the **Mode Shape (solid2)** toolbar, click **Plot**.
6 Click the **Show Grid** button in the **Graphics** toolbar.

   Also twisting modes can be displayed.
7 From the **Eigenfrequency (Hz)** list, choose **155.04**.
8 In the **Mode Shape (solid2)** toolbar, click **Plot**.

SOLID MECHANICS (SOLID)

The twisting modes can also be computed using the axisymmetric Solid Mechanics interface. To do that, use circumferential mode extension.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
2 In the **Settings** window for **Solid Mechanics**, locate the **Axial Symmetry Approximation** section.
3 Select the **Circumferential mode extension (time-harmonic)** check box.

STUDY 1, 2D AXISYMMETRIC SOLID

Step 1: Eigenfrequency

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

2 Find the **Physics and Variable Selection** subsection. In the table, clear the **Solve** check box for **Shell** and **Solid Mechanics 2** interface.
3 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
4 Select the **Desired number of eigenfrequencies** check box.
5 In the associated text field, type `10`.
6 In the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape, 3D (solid)

In the **Model Builder** window, expand the **Results>Mode Shape, 3D (solid)** node.

Deformation

To display also twisting modes, add the rotational displacement component to the deformation.

1 In the **Model Builder** window, expand the **Results>Mode Shape, 3D (solid)>Surface 1** node, then click **Deformation**.
2 In the **Settings** window for **Deformation**, locate the **Expression** section.
3 In the **PHI component** text field, type `v`.

Mode Shape, 3D (solid)

Display the first twist mode.

1 In the **Model Builder** window, 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 **155.04**.
4 In the **Mode Shape, 3D (solid)** toolbar, click **Plot**.
5 From the **Eigenfrequency (Hz)** list, choose **243.5**.
6 In the **Mode Shape, 3D (solid)** toolbar, click **Plot**.

STUDY 2, 2D AXISYMMETRIC SHELL

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 2, 2D axisymmetric shell** click **Step 1: Eigenfrequency**.
2 Find the **Physics and Variable Selection** subsection. In the table, clear the **Solve** check box for **Solid Mechanics 2** interface.

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.

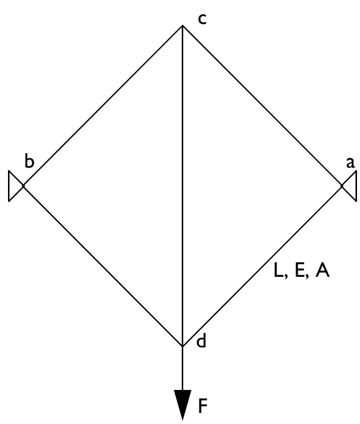


Figure 1: The truss geometry.

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 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 case, 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	$-5.14 \cdot 10^{-4}$ m	$-5.15 \cdot 10^{-4}$ m
Displacement at c	$-2.13 \cdot 10^{-4}$ m	$-2.13 \cdot 10^{-4}$ 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.

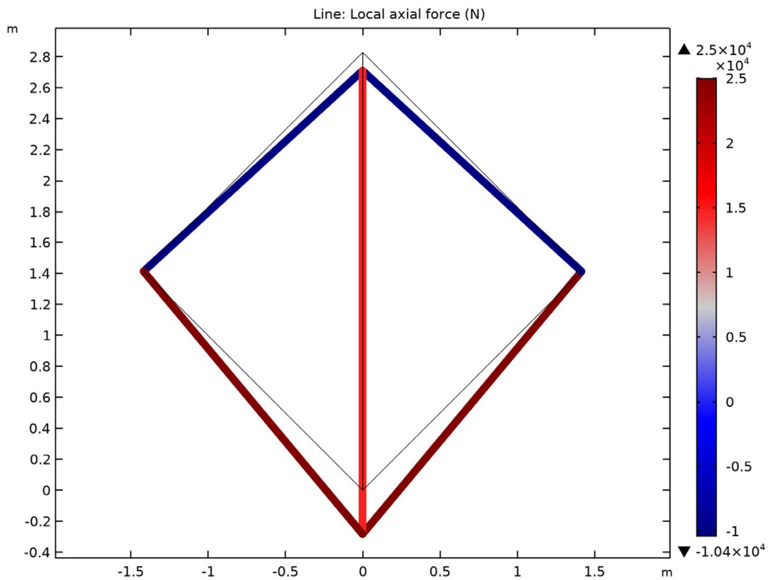


Figure 2: Deformed geometry and axial forces for the 2D case.

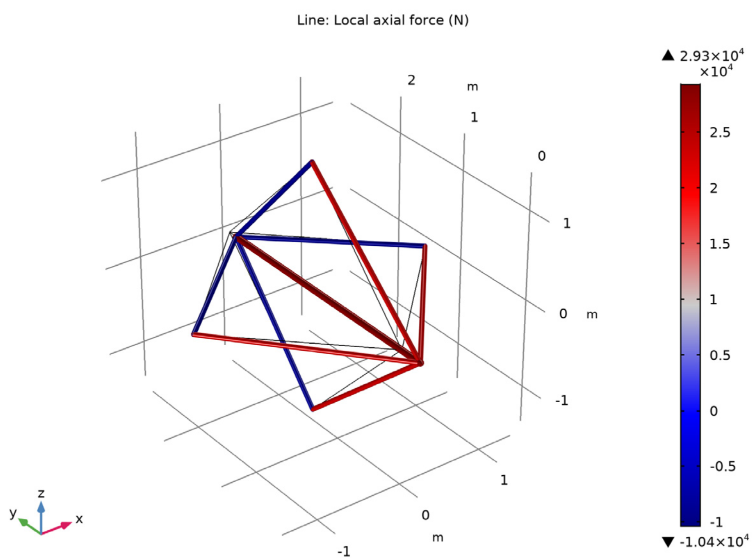


Figure 3: Deformed geometry and axial forces for the 3D case.

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 **General Studies>Stationary**.
6. Click **Done**.

GEOMETRY 1

Square 1 (sq1)

1. In the **Geometry** toolbar, click **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 in the **Graphics** toolbar.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

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

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 In the **y** text field, type `sqrt(8)`.

6 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 In the **Physics** toolbar, click **Points** and choose **Pinned**.

2 Select Points 1 and 4 only.

Point Load 1

1 In 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
-50e3	y

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	**Variable**	**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

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

RESULTS

Force (truss)

1 In the **Model Builder** window, expand the **Results>Force (truss)** node, then click **Force (truss)**.

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

3 Select the **Show maximum and minimum values** check box.

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

Derived Values

Next, compute the displacements at d (Vertex 2) and c (Vertex 3).

Point Evaluation 1

1 In 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 - m>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 nonlocal average couplings for the members ac, ad, and cd. You will use these for defining variables that evaluate the axial forces in these members.

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal 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 In the **Definitions** toolbar, click **Nonlocal 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 In the **Definitions** toolbar, click **Nonlocal 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 In 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 **Study 1>Solver Configurations>Solution 1 (sol1)** and choose **Solution>Update**.

RESULTS

Global Evaluation 1

1 In 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.

TABLE

1 Go to the **Table** window.

Now create the 3D truss as a new model.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

ADD PHYSICS

1 In 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 2** in the window toolbar.

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

ROOT

In the **Model Builder** window, click the root node.

ADD STUDY

1 In 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 **General 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 From the **Home** menu, choose **Add Study**.

GEOMETRY 2

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

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

1 Right-click **Work Plane 1 (wp1)** and choose **Show Work Plane**.

2 In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Square 1 (sq1)

1 In the **Work Plane** toolbar, click **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 In the **Work Plane** toolbar, click **Build All**.

7 In the **Model Builder** window, click **Geometry 2**.

Rotate 1 (rot1)

1 In 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 **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 In the **Angle** text field, type `90`.

9 Click **Build All Objects**.

10 In the **Model Builder** window, click **Geometry 2**.

Line Segment 1 (ls1)

1. In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.
2. In the **Settings** window for **Line Segment**, locate the **Starting Point** section.
3. From the **Specify** list, choose **Coordinates**.
4. Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
5. In the **y** text field, type `sqrt(8)`.
6. Click **Build All Objects**.

DEFINITIONS (COMP2)

Add nonlocal average couplings for the members ac, ad, and cd and corresponding axial force variables.

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

Average 4 (aveop4)

1. In the **Definitions** toolbar, click **Nonlocal 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.
5. In the **Model Builder** window, under **Component 2 (comp2)** click **Definitions**.

Average 5 (aveop5)

1. In the **Definitions** toolbar, click **Nonlocal 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.
5. In the **Model Builder** window, click **Definitions**.

Average 6 (aveop6)

1. In the **Definitions** toolbar, click **Nonlocal 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.
5. In the **Model Builder** window, click **Definitions**.

Variables 2

1. In 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, under **Component 2 (comp2)>Truss 2 (truss2)** 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. In 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. In the **Physics** toolbar, click **Points** and choose **Pinned**.
2. Select Points 1, 3, 4, and 6 only.

Point Load 1

1. In 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
-100e3	y
0	z

MATERIALS

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 2 (comp2)** 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	Variable	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

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

RESULTS

Force (truss2)

1 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

2 Select the **Show maximum and minimum values** check box.

Derived Values

Proceed to compute the displacements at d (Vertex 2) and c (Vertex 5).

Point Evaluation 2

1 In the **Model Builder** window, right-click **Derived Values** and choose **Point Evaluation**.

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

3 From the **Dataset** 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 - m>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

Derived Values

In the **Model Builder** window, under **Results** click **Derived Values**.

Global Evaluation 2

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** 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 5.5

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.

In addition, it is shown how to evaluate modal participation factors and modal masses.

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).

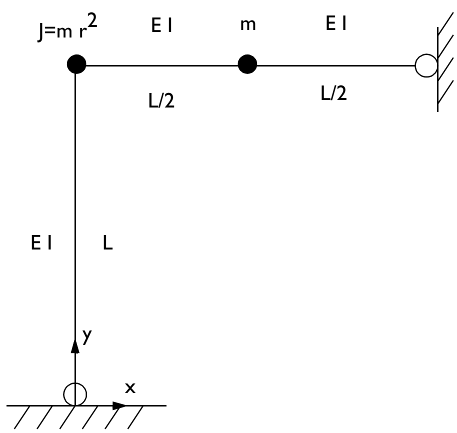


Figure 1: Definition of the problem.

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 $A = 9\cdot10^{-4}$ 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 $L/4$. This gives the value 62.5 kgm^2.

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.

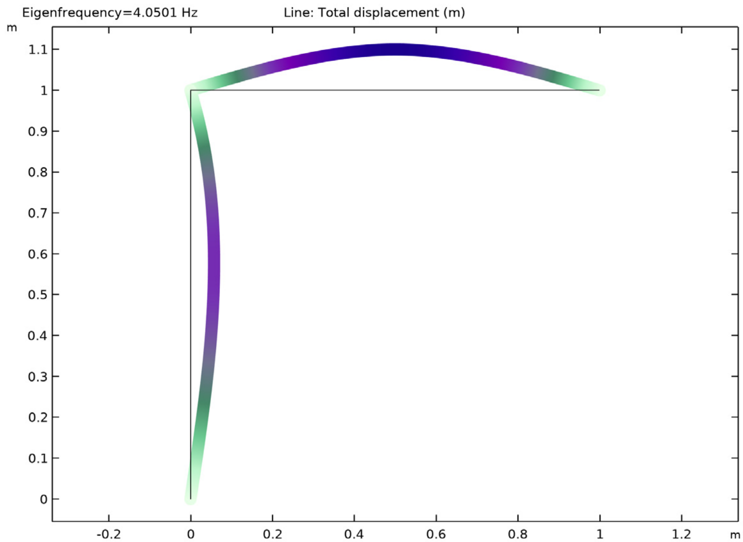


Figure 2: The first eigenmode.

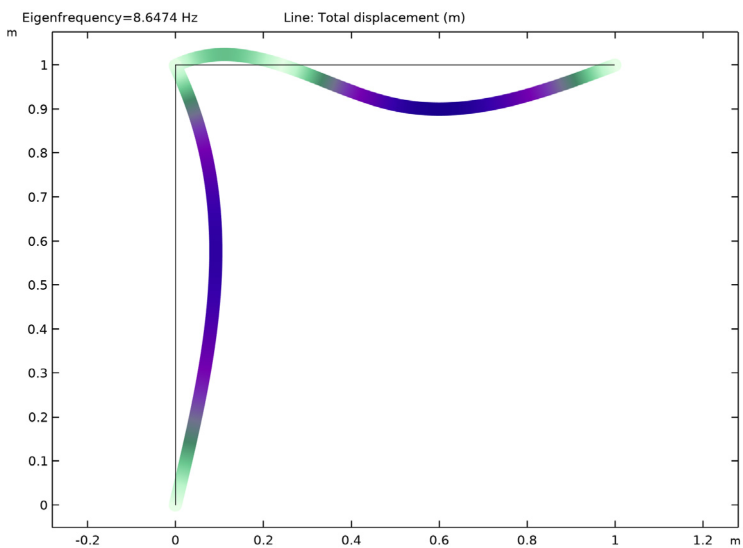


Figure 3: The second eigenmode.

Because the beams have no density in this example, the total mass is the 1000 kg supplied by the point mass. The mass moment of inertia is also a point contribution, and has the value 62.5 kgm^2. The mass represented by the computed eigenmodes can be evaluated using the modal participation factors, see Figure 4 and Figure 5. 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. Similarly, all rotational inertia is captured by the first two modes.

Eigenfrequency (Hz)	Participation factor, normalized, X-translation (1)	Participation factor, normalized, Y-translation (1)	Effective modal mass, Y-translation (kg)
4.0501	-0.0085424	25.314	640.78
8.6474	-0.011483	-18.953	359.22

Figure 4: Participation factors for each eigenfrequency.

Integral: Effective modal mass, Y-translation (kg)	Integral: Effective modal mass, Z-rotation (kg*m^2)
1000.0	62.500

Figure 5: Summed modal masses.

Notes About the COMSOL Implementation

The variables for evaluation of participation factors are created in the **Participation Factors** node under **Definitions**. This node is created automatically when an **Eigenfrequency** study is added.

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 **General Studies>Eigenfrequency**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `inplane_framework_freq_parameters.txt`.

GEOMETRY 1

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **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 table, enter the following settings:

x (m)	y (m)
0	0
0	L
L/2	L
L	L

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	Variable	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, 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 `a`.

5 In the h_z text field, type `a`.

Pinned 1

1 In the **Physics** toolbar, click **Points** and choose **Pinned**.

2 Select Points 1 and 4 only.

Point Mass 1

1 In 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 In 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, under **Study 1** 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`.

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

RESULTS

Line 1

1 In the **Model Builder** window, expand the **Results>Mode Shape (beam)** node, then click **Line 1**.

2 In the **Mode Shape (beam)** toolbar, click **Plot**.

3 Click the **Zoom Extents** button in the **Graphics** toolbar.

Mode Shape (beam)

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

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

3 From the **Eigenfrequency (Hz)** list, choose **8.6474**.

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

Derived Values

Compare the computed eigenfrequencies to the analytical values.

Global Evaluation 1

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Eigenfrequnecy comparison` in the **Label** text field.

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

Expression	Unit	Description
`f1`	`1/s`	`Eigenfrequency 1, analytical`
`f2`	`1/s`	`Eigenfrequency 2, analytical`

4 Click **Evaluate**.

Participation factors (Study 1)

Examine the modal participation factors.

Finally, compute the total effective mass accounted for in the computed eigenmodes.

Global Evaluation 2

1. In the **Results** toolbar, click **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, type `Summed modal masses` in the **Label** text field.
3. Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Definitions>Participation Factors 1>Effective modal mass> mpf1.mEffLY - Effective modal mass, Y-translation - kg**.
4. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`mpf1.mEffLY`	`kg`	`Effective modal mass, Y-translation`
`mpf1.mEffRZ`	`kg*m^2`	`Effective modal mass, Z-rotation`

5. Locate the **Data Series Operation** section. From the **Operation** list, choose **Integral**.
6. From the **Method** list, choose **Summation**.
7. Click **Evaluate**.

Created in COMSOL Multiphysics 5.5

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.

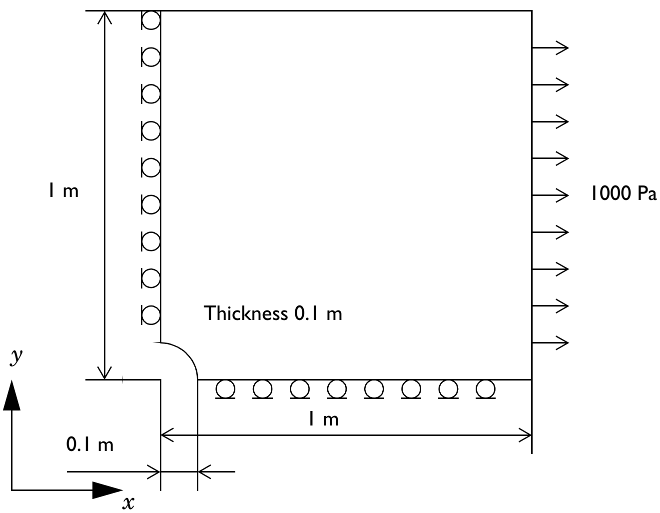


Figure 1: Geometry model of the Kirsch plate with rollers defining the symmetry plane.

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 and Discussion

The distribution of the normal stress in the x direction, σ_x, is shown in Figure 2and Figure 3. The stress contours of the finite model and the infinite model are very similar.

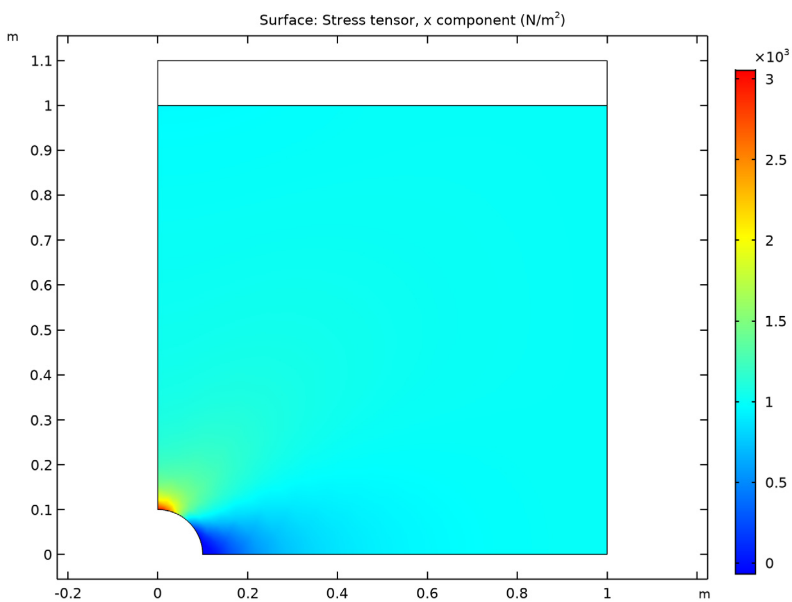


Figure 2: Distribution of the normal stress in the x direction for the finite model.

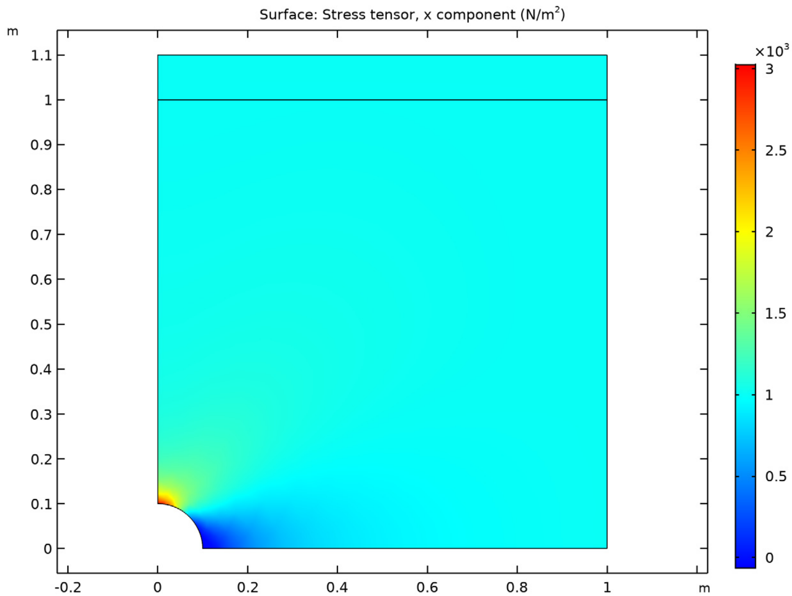


Figure 3: Distribution of the normal stress in the x direction for the infinite model.

According to Ref. 1 the stress σ_x along the vertical symmetry line can be calculated as

$$\sigma_x = \frac{1000}{2}\left(2 + \frac{0{,}1^2}{y^2} + 3\frac{0{,}1^4}{y^4}\right) \tag{1}$$

Figure 4 shows the stress σ_x obtained from the solved models, and plotted as a function of the true y-coordinate along the left symmetry edge, which are in close agreement with the theoretical value according to Equation 1.

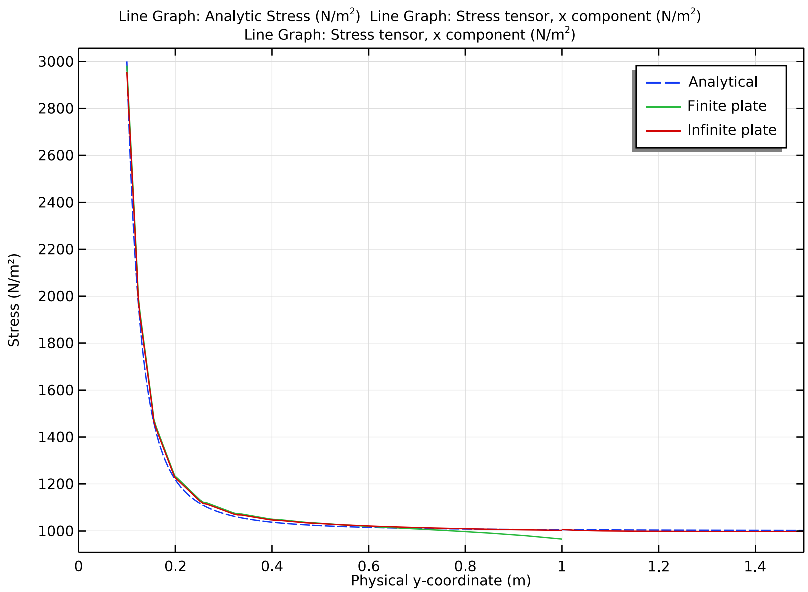


Figure 4: Normal stress, simulated results (solid line) versus the theoretical values (dashed line).

Away from the hole, stresses from the finite model starts drifting from the theoretical values, while stresses from the infinite model matches closely with the theoretical value.

The stress error is reported in the following table:

TABLE 1: STRESS ERROR RELATIVE TO ANALYTICAL SOLUTION.

	FINITE PLATE	INFINITE PLATE
Near hole	1.1 %	0.1 %
Away from hole	-4 %	-0.1 %

Notes About the COMSOL Implementation

The default scaling function in **Infinite Element Domain** is rational. This type of function is well adapted to cases where the degrees of freedom vanish to zero at infinity. The present model is submitted to infinite loads at infinity, that means that constant strain and linear displacement are expected. For this type of infinite solution, polynomial functions are preferred. The relation between the stretched and geometric coordinates is

$$X_{\mathrm{m}} - X_0 = f\left(\frac{X - X_0}{\Delta X}\right)$$

where the function f is defined with an analytic function. Here we want f as a second-order polynomial: $f(\xi) = a\xi^2 + b\xi + c$. The continuity condition at X0, $f(0) = 0$, and at the end of the domain $f(1) = p_{\mathrm{w}}$ imply that the polynomial is:

$$f(\xi) = (p_{\mathrm{w}} - b)\xi^2 + b\xi$$

The infinite element domain gives best results when meshed with rectangular elements; see Figure 5.

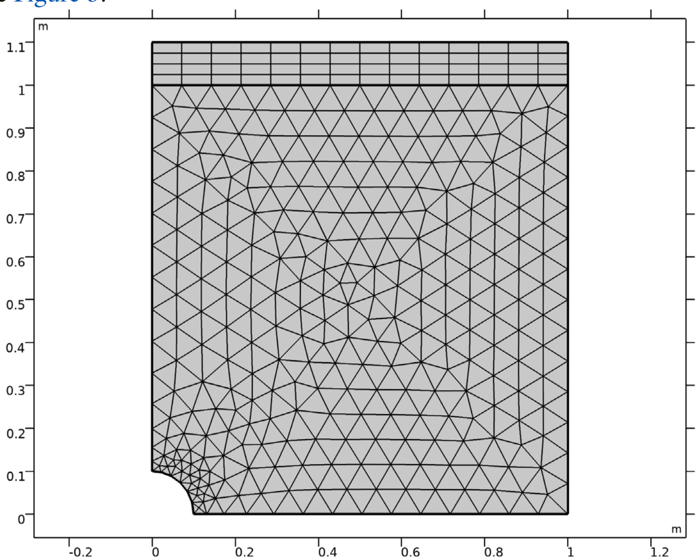


Figure 5: Infinite element domain modeled with rectangular elements.

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 **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
pw	10[m]	10 m	Physical width of infinite element domain
deltaY	0.1[m]	0.1 m	Geometric thickness of infinite element layer

Draw a rectangle with a top layer that represents the infinite element domain.

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

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

3 In the **Height** text field, type `1+deltaY`.

4 Click to expand the **Layers** section. Clear the **Layers on bottom** check box.

5 Select the **Layers on top** check box.

6 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	deltaY

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

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

3 In the **Radius** text field, type `0.1`.

4 Click **Build Selected**.

Difference 1 (dif1)

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

2 Add the rectangle and remove the circle in the **Difference** section.

3 In the **Settings** window for **Difference**, click **Build All Objects**.

GLOBAL DEFINITIONS

First add an analytical function for stress, based on Kirsch's theoretical solution of an infinite plate.

Analytic 1 (an1)

1 In the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

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

3 In the **Function name** text field, type `AnaStress`.

4 Locate the **Definition** section. In the **Expression** text field, type `1000/2*(2+(0.1/y)^2+3*(0.1/y)^4)`.

5 In the **Arguments** text field, type `y`.

6 Locate the **Units** section. In the **Arguments** text field, type `m`.

7 In the **Function** text field, type `N/m^2`.

Create an analytic polynomial function to define the scaling in the infinite element domain.

DEFINITIONS

Analytic 2 (an2)

1 In the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, locate the **Definition** section.

3 In the **Expression** text field, type `(pw-10*deltaY)*x^2+10*deltaY*x`.

4 Locate the **Units** section. In the **Arguments** text field, type `m`.

5 In the **Function** text field, type `m`.

Infinite Element Domain 1 (ie1)

1 In the **Definitions** toolbar, click **Infinite Element Domain**.

2 Select Domain 2 only.

3 In the **Settings** window for **Infinite Element Domain**, locate the **Scaling** section.

4 From the **Coordinate stretching type** list, choose **User defined**.

5 From the **Stretching function** list, choose **Analytic 2 (an2)**.

Add a variable representing the physical y-coordinate to be used in postprocessing.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
ym	if(dom==2,ie1.Ym,y)	m	Physical y-coordinate

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 (infinite elements)**.

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. In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2. Select Boundaries 1, 2, and 5 only.

Boundary Load 1

1. In 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. In 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 1** in the window toolbar.
5. In the **Model Builder** window, click **Component 1 (comp1)**.
6. In the **Physics** toolbar, click **Add Physics** to close the **Add Physics** window.

SOLID MECHANICS 2 (SOLID2)

1. In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.
2. From the list, choose **Plane stress**.
3. Locate the **Thickness** section. In the d text field, type `0.1`.

Symmetry 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2. Select Boundaries 1, 2, and 5 only.

Boundary Load 1

1. In 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

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	Variable	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 lower element size is set at the expected location of stress concentration.

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 1 only.

Size 1

1 Right-click **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 **Point**.

4 Select Point 1 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 `0.02`.

Infinite elements give better results when meshed with rectangular elements.

Mapped 1

In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

Distribution 1

1 In the **Model Builder** window, 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 `4`.

5 Click **Build All**.

STUDY 1

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

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

3 Select Points 1 and 2 only.

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

Expression	Unit	Description
`(solid.sx-AnaStress(y))/AnaStress(y)`		`Error in finite plate`
`(solid2.sx-AnaStress(y))/AnaStress(y)`		`Error in infinite plate`

5 Click **Evaluate**.

Stress (solid)

The default plots show the von Mises stress combined with a scaled deformation of the plate. Remove deformation and 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) - N/m²>solid.sx - Stress tensor, x component**.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node.

2 Right-click **Deformation** and choose **Delete**.

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) - N/m²>solid2.sx - Stress tensor, x component**.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node.

2 Right-click **Deformation** and choose **Delete**.

1D Plot Group 5

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Stress Profile` in the **Label** text field.

Line Graph 1

1 Right-click **Stress Profile** and choose **Line Graph**.

2 Select Boundaries 1 and 2 only.

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

4 In the **Expression** text field, type `AnaStress(ym)`.

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

6 In the **Expression** text field, type `ym`.

7 In the **Stress Profile** toolbar, click **Plot**.

8 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

9 Select the **Show legends** check box.

10 In the table, enter the following settings:

Legends
`Analytical`

11 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

12 In the **Stress Profile** toolbar, click **Plot**.

Line Graph 2

1 In the **Model Builder** window, right-click **Stress Profile** 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) - N/m²>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 Right-click **Stress Profile** and choose **Line Graph**.

2 Select Boundaries 1 and 2 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) - N/m²>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 `ym`.

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`

Stress Profile

1 In the **Model Builder** window, click **Stress Profile**.

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

3 Select the **Manual axis limits** check box.

4 In the **x minimum** text field, type `0`.

5 In the **x maximum** text field, type `1.5`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `Physical y-coordinate (m)`.

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

9 In the associated text field, type `Stress (N/m²)`.

10 In the **Stress Profile** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.5

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.

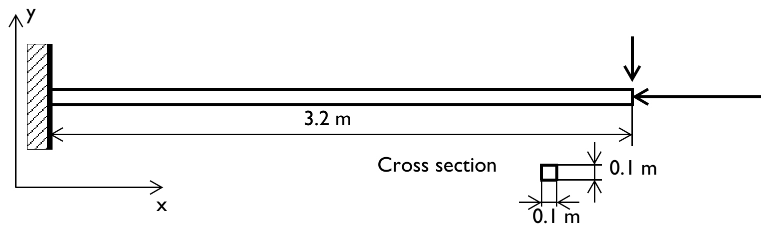


Figure 1: Cantilever beam geometry.

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 $E = 2.1 \cdot 10^{11}$ N/m^2 and $\nu = 0$.

CONSTRAINTS AND LOADS

- The left end is fixed.
- The right end is subjected to a 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 the results are compared with the benchmark value. Using the Solid Mechanics interface, the problem is modeled as a "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. The load on the right end of the beam is modeled as a uniformly distributed boundary load, corresponding to the specified total load.

In the second part of this problem, a linear buckling analysis study is carried out to compute the critical buckling load of the structure.

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.

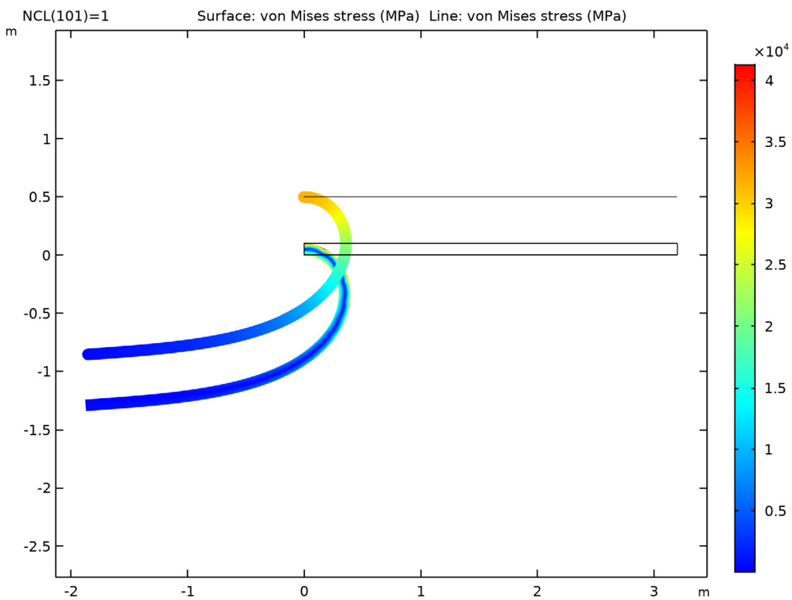


Figure 2: The effective von Mises stress of the deformed beam.

The horizontal and vertical displacements of the tip versus the compressive load normalized by its maximum value are shown in Figure 3.

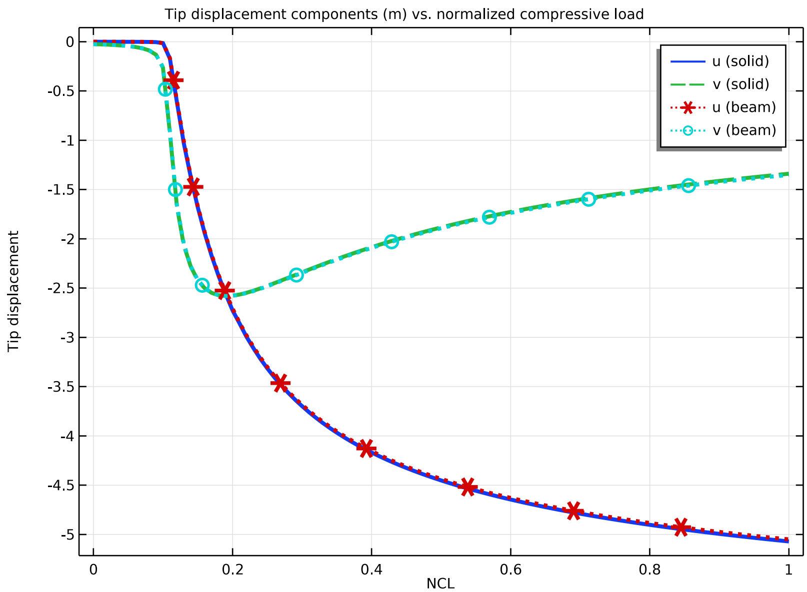


Figure 3: Horizontal and vertical tip displacements versus normalized compressive load.

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

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

Figure 4 shows the first buckling mode of the beam computed from a linear buckling analysis.

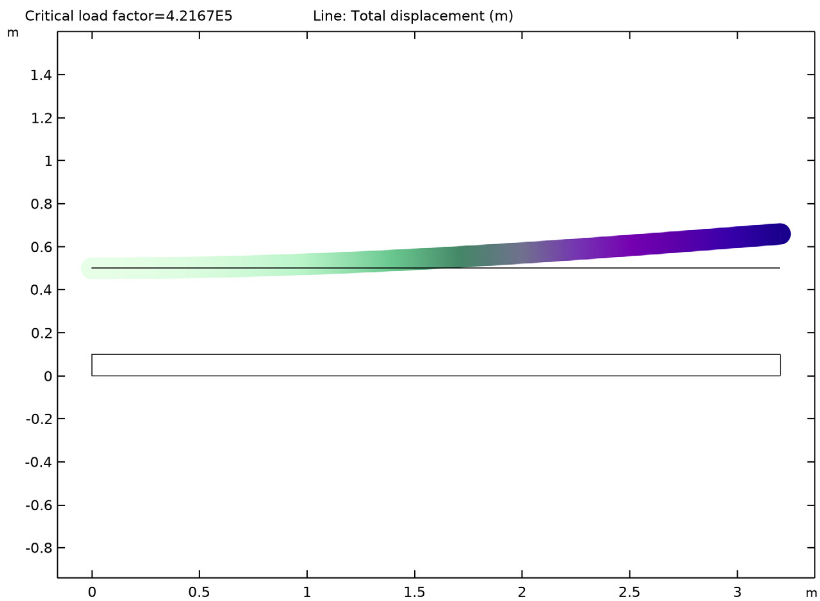


Figure 4: First buckling mode of the beam.

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 **General Studies>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

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `large_deformation_beam_parameters.txt`.

   By restricting the range of parameter **NCL** to [0, 1], it serves as a compressive load normalized by maximum compressive load.

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **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`.

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

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

3 In the table, enter the following settings:

x (m)	y (m)
0	5*d
1	5*d

4 Click **Build All Objects**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 Click **Build Selected**.

GLOBAL DEFINITIONS

In this example, the same material data will be referenced for **Solid Mechanics** and **Beam** interfaces, hence it can be added as a **Global Material** in the model. Using **Material Link** node, we assign the **Global Material** to different domains, boundaries and edges of the structure.

Material 1 (mat1)

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, click to expand the **Material Properties** section.

3 In the **Material properties** tree, select **Basic Properties>Density**.

4 Click **Add to Material**.

5 In the **Material properties** tree, select **Basic Properties>Poisson's Ratio**.

6 Click **Add to Material**.

7 In the **Material properties** tree, select **Basic Properties>Young's Modulus**.

8 Click **Add to Material**.

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

Property	Variable	Value	Unit	Property group
Density	rho	7850	kg/m³	Basic
Poisson's ratio	nu	0	1	Basic
Young's modulus	E	2.1e5[MPa]	Pa	Basic

MATERIALS

Material Link 1 (matlnk1)

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

Material Link 2 (matlnk2)

1 Right-click **Materials** and choose **More>Material Link**.
2 In the **Settings** window for **Material Link**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 4 only.

Add physics settings for the **Solid Mechanics** interface.

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 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
2 Select Boundary 1 only.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** 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 $\mathbf{F}_{\mathrm{tot}}$ 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 **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 Select Point 3 only.

Point Load 1

1 In the **Physics** toolbar, click **Points** and choose **Point Load**.

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

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

NCL*F_Lx	x
F_Ly	y

4 Select Point 6 only.

Add unit point load for linear buckling analysis.

Point Load 2

1 Right-click **Point Load 1** and choose **Duplicate**.

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

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

-1	x
0	y

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 **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 In the **Model Builder** window, 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 Click **Build All**.

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 **Study Settings** section.

3 Select the **Include geometric nonlinearity** check box.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Beam (beam)>Point Load 2**.

6 Click **Disable**.

7 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

8 Click **Add**.

9 In the table, enter the following settings:

Parameter name	**Parameter value list**
NCL (Normalized compressive load)	`range(0,0.01,1)`

10 Right-click **Study 1>Step 1: Stationary** and choose **Get Initial Value for Step**.

STUDY 1

Solver Configurations

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 **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 Under **Variables**, click **Delete**.

13 Click to expand the **Method and Termination** section. From the **Termination technique** list, choose **Tolerance**.

14 In the **Model Builder** window, right-click **Segregated 1** and choose **Segregated Step**.

15 In the **Settings** window for **Segregated Step**, locate the **General** section.

16 Under **Variables**, click **Add**.

17 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)**.

18 Click **OK**.

19 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

20 From the **Nonlinear method** list, choose **Automatic (Newton)**.

21 In the **Maximum number of iterations** text field, type `200`.

22 In the **Tolerance factor** text field, type `1`.

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Stress (beam)**.

5 In 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 **Line 1** and choose **Copy**.

Line 1

1 In the **Model Builder** window, 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, 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, click **Stress (solid and beam)**.

2 In the **Settings** window for **Line**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Surface 1**.

4 Clear the **Tube radius scale factor** check box.

Stress (solid)

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

2 In the **Stress (solid)** toolbar, click **Plot**.

3 Click the **Zoom Extents** button in 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 In 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 `l`.

4 In the **Y** text field, type `d/2`.

5 Click **Plot**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

1D Plot Group 8

1 In 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 **Dataset** 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 - m>u - Displacement field, X component**.

3 Click to expand 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, 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 - m>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 **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
4 Locate the **Selection** section. Select the **Activate selection** 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 - m>u2 - Displacement field, X component**.
7 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 **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 - m>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 In the **Tip displacement** toolbar, click **Plot**.

Tip displacement

1 In the **Model Builder** window, 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 In the **Tip displacement** toolbar, click **Plot**.

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

Evaluate the deformation of the structure.

Point Evaluation 1

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

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

3 From the **Dataset** 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 - m>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`

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 **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 Locate the **Selection** section. Select the **Activate selection** 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`
`uFinal_Ref`	`m`	`Reference value for final horizontal displacement at the tip`

7 Click **Table 1 - Point Evaluation 1**.

Point Evaluation 3

1 In the **Model Builder** window, under **Results>Derived Values** right-click **Point Evaluation 1** and choose **Duplicate**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`v`	`m`	`Solid: y-disp`

4 Click **New Table**.

Point Evaluation 4

1 In the **Model Builder** window, under **Results>Derived Values** right-click **Point Evaluation 2** and choose **Duplicate**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`v2`	`m`	`Beam: y-disp`
`vFinal_Ref`	`m`	`Reference value for final vertical displacement at the tip`

4 Click **Table 2 - Point Evaluation 3**.

Point Evaluation 5

1 In the **Model Builder** window, under **Results>Derived Values** right-click **Point Evaluation 3** and choose **Duplicate**.
2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.
3 From the **Parameter selection (NCL)** list, choose **All**.
4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
abs(v)	m	Solid: y-disp

5 Locate the **Data Series Operation** section. From the **Operation** list, choose **Maximum**.
6 Click **New Table**.

Point Evaluation 6

1 In the **Model Builder** window, under **Results>Derived Values** right-click **Point Evaluation 4** and choose **Duplicate**.
2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.
3 From the **Parameter selection (NCL)** list, choose **All**.
4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
abs(v2)	m	Beam: y-disp
abs(vMax_Ref)	m	

5 Locate the **Data Series Operation** section. From the **Operation** list, choose **Maximum**.
6 Click **Table 3 - Point Evaluation 5**.

ADD STUDY

1 In 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 for Selected Physics Interfaces>Linear Buckling**.
4 Click **Add Study** in the window toolbar.
5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

1. In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
2. Select the **Modify model configuration for study step** check box.
3. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)**.
4. Click **Disable**.
5. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Beam (beam)>Point Load 1**.
6. Click **Disable**.

Step 2: Linear Buckling

1. In the **Model Builder** window, click **Step 2: Linear Buckling**.
2. In the **Settings** window for **Linear Buckling**, locate the **Physics and Variables Selection** section.
3. Select the **Modify model configuration for study step** check box.
4. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)**.
5. Click **Disable**.
6. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Beam (beam)>Point Load 1**.
7. Click **Disable**.
8. In the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (beam)

Click the **Zoom Extents** button in the **Graphics** toolbar.

Point Evaluation 7

1. In the **Results** toolbar, click **Point Evaluation**.
2. In the **Settings** window for **Point Evaluation**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
4. Select Point 6 only.

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

Expression	Unit	Description
`Fcr`	`N`	`First critical buckling load`

6 Click **Evaluate**.

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.

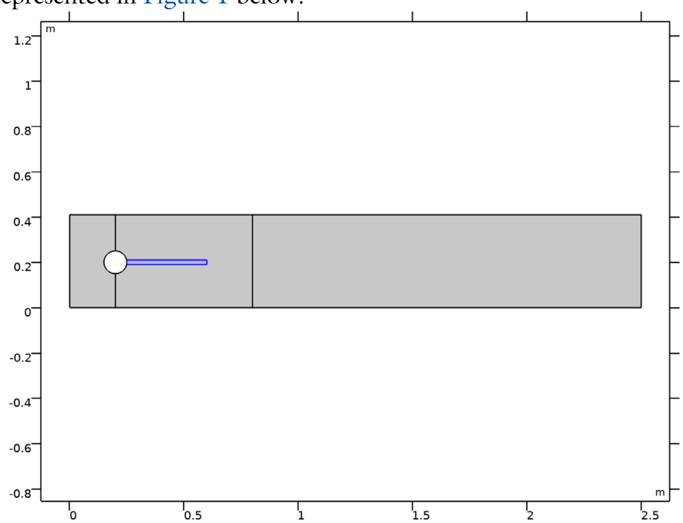


Figure 1: Model geometry including solid and fluid domains (blue and gray, respectively).

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 the 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.

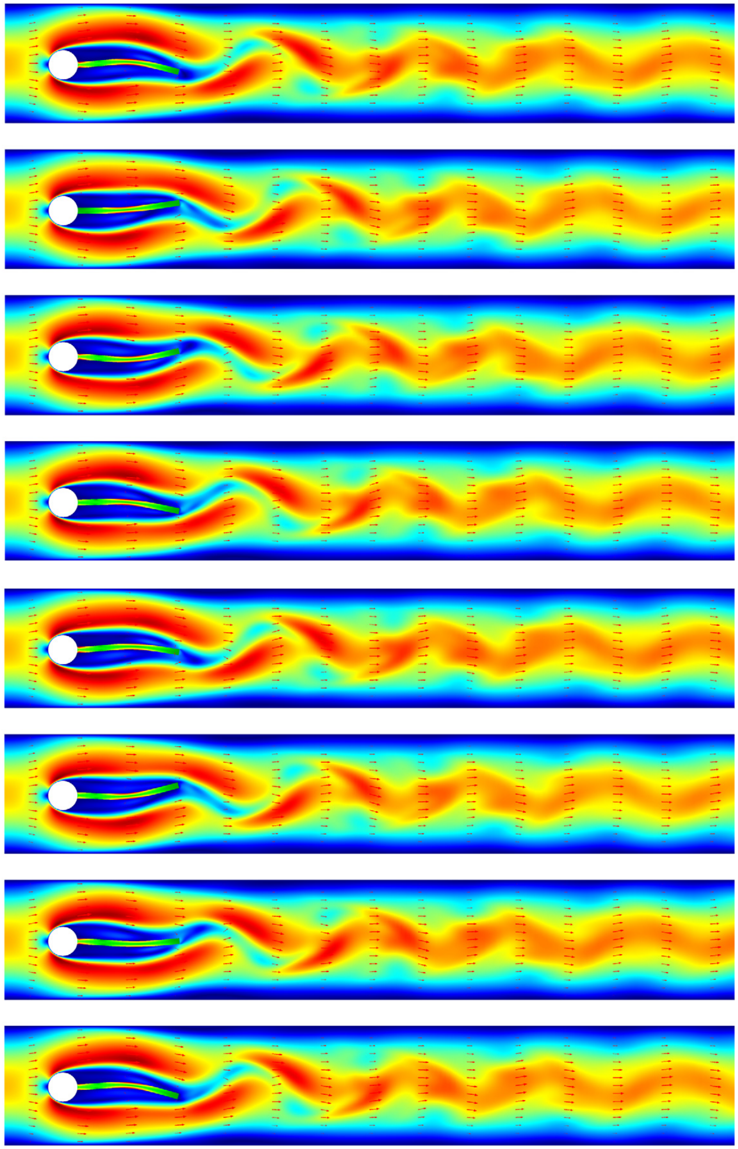


Figure 2: Velocity field in fluid and von Mises stress in structure for eight different time steps.

[Figure 3](#) below shows the evolution of the fluid forces all along the time step. The oscillation is 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.

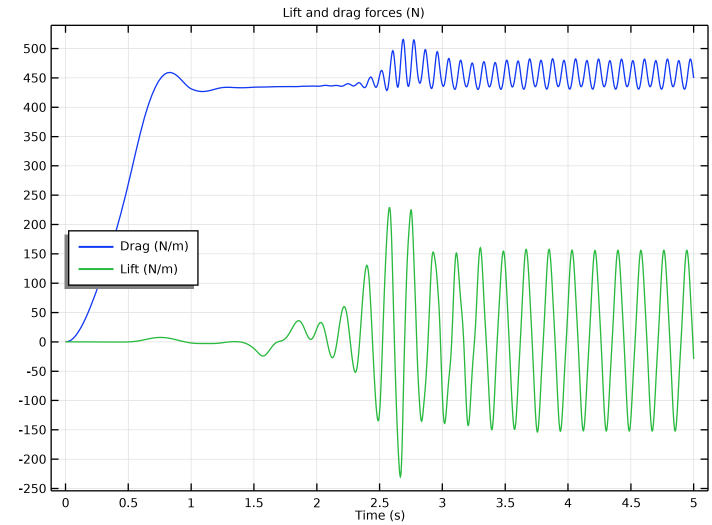


Figure 3: Drag and lift forces versus time.

[Figure 4](#) shows the displacement of the tip of the beam in the x and y directions:

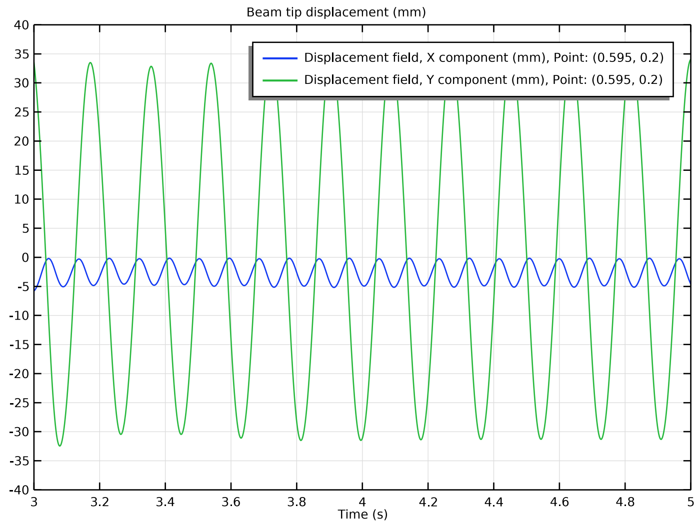


Figure 4: Tip displacement of the structure in the x and y directions (in green and blue respectively).

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 1 mm with an oscillation magnitude of 33 mm, in good agreement with the targeted value.

The trajectory of the tip is shown in Figure 6.

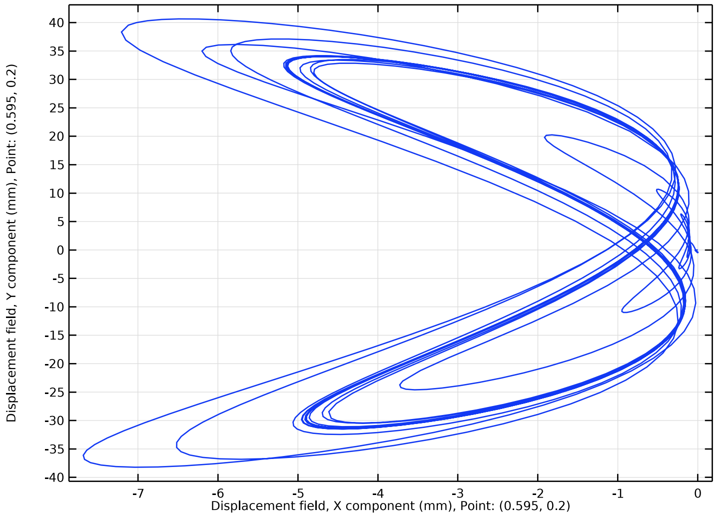


Figure 5: Beam tip trajectory. The origin corresponds to the initial position.

Figure 6 below shows the frequency spectrum of the structure oscillation.

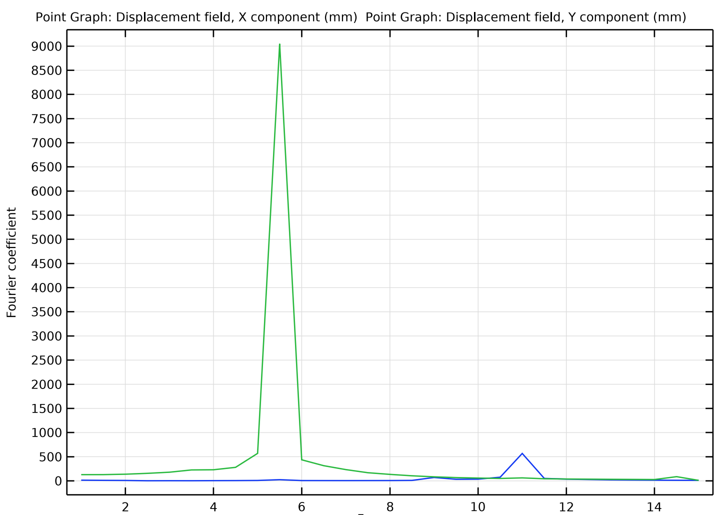


Figure 6: Frequency spectrum of the structure tip displacement.

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 5.5 Hz, which agree well with the targeted results.

Figure 7 below shows the variations of the lift and drag forces applied to the structure:

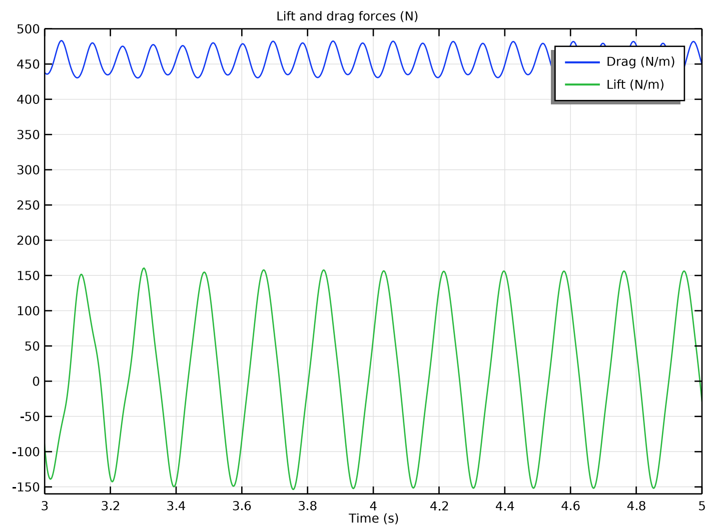


Figure 7: Lift and drag forces (green and blue curves, respectively) after the periodic oscillations have established.

The average of the total lift force is about 2 N with an oscillation magnitude of 154 N, while the drag force average is about 456 N with an oscillation magnitude of 26 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, use P2+P2 elements to increase the accuracy for the flow equations.

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>Fluid-Solid Interaction**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Time Dependent**.
6 Click **Done**.

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **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 In the **Geometry** toolbar, click **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 In the **Geometry** toolbar, click **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`.

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **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 `0.41`.

5 Locate the **Position** section. In the **x** text field, type `0.2`.

Difference 1 (dif1)

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

2 In the **Settings** window for **Difference**, type `Solid` in the **Label** text field.

3 Select the object **r2** only.

4 Locate the **Difference** section. Find the **Objects to subtract** subsection. Select the **Activate selection** toggle button.

5 Select the object **c1** only.

6 Select the **Keep input objects** check box.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Difference 2 (dif2)

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

2 In the **Settings** window for **Difference**, type `Fluid` in the **Label** text field.

3 Select the objects **r1** and **r3** only.

4 Locate the **Difference** section. Find the **Objects to subtract** subsection. Select the **Activate selection** toggle button.

5 Select the objects **c1** and **r2** only.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Form Union (fin)

1 In the **Geometry** toolbar, click **Build All**.

2 In the **Model Builder** window, click **Form Union (fin)**.

3 Click **Build Selected**.

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

DEFINITIONS

Deforming Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Moving Mesh** click **Deforming Domain 1**.

2 Select Domain 2 only.

Step 1 (step1)

1 In the **Home** toolbar, click **Functions** and choose **Global>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 In the **Home** toolbar, click **Functions** and choose **Global>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`.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Fluid**.

4 Click the **Show More Options** button in the **Model Builder** toolbar.

5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Stabilization**.

6 Click **OK**.

7 In the **Model Builder** window, click **Laminar Flow (spf)**.

8 In the **Settings** window for **Laminar Flow**, click to expand the **Consistent Stabilization** section.

9 Find the **Navier-Stokes equations** subsection. Clear the **Crosswind diffusion** check box.

10 Click to expand the **Discretization** section. From the **Discretization of fluids** list, choose **P2+P2**.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **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 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 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 **Domain Selection** section.

3 From the **Selection** list, choose **Solid**.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundaries 19 and 20 only.

Point Load 1

1 In the **Physics** toolbar, click **Points** and choose **Point Load**.

2 Select Point 11 only.

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

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

0	x
1[N]*gp1(t/1[s])	y

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 **Solid**.

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

Property	**Variable**	**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 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Fluid**.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Density	rho	`1000`	kg/m³	Basic
Dynamic viscosity	mu	`1`	Pa·s	Basic

MESH 1

Size

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

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

3 From the **Predefined** list, choose **Coarse**.

Size 1

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

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

3 From the **Predefined** list, choose **Normal**.

Free Triangular 1

1 In the **Model Builder** window, click **Free Triangular 1**.
2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domains 1–3 only.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.
2 Right-click **Mapped 1** and choose **Move Up**.
3 In the **Settings** window for **Mapped**, click to expand the **Control Entities** section.
4 Clear the **Smooth across removed control entities** check box.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 12 and 13 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 From the **Distribution type** list, choose **Predefined**.
5 In the **Number of elements** text field, type 40.
6 In the **Element ratio** text field, type 5.
7 Select the **Reverse direction** check box.

Boundary Layers 1

1 In the **Model Builder** window, click **Boundary Layers 1**.
2 In the **Settings** window for **Boundary Layers**, click to expand the **Corner Settings** section.
3 From the **Handling of sharp corners** list, choose **None**.
4 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** check box.
5 Click **Build All**.

   You can now prepare the probe variables to display during the computation.

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal 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 8–10 and 15–18 only.

Global Variable Probe 1 (var1)

1 In 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(spf.T_stressx)`.

4 Select the **Description** check box.

5 In the associated text field, type `Drag`.

6 Click to expand the **Table and Window Settings** section. Click **Add Plot Window**.

Global Variable Probe 2 (var2)

1 Right-click **Global Variable Probe 1 (var1)** and choose **Duplicate**.

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(spf.T_stressy)`.

4 In the **Description** text field, type `Lift`.

Domain Point Probe 1

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.

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

3 From the **Frame** list, choose **Material**.

4 In row **Coordinates**, set **X** to `0.595`.

5 In row **Coordinates**, set **Y** to `0.2`.

Point Probe Expression 1 (ppb1)

1 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

2 In the **Settings** window for **Point Probe Expression**, type `u` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `u_solid`.

4 From the **Table and plot unit** list, choose **mm**.

5 Click to expand the **Table and Window Settings** section. Click **Add Plot Window**.

Point Probe Expression 2 (ppb2)

1 Right-click **Component 1 (comp1)>Definitions>Domain Point Probe 1> Point Probe Expression 1 (ppb1)** and choose **Duplicate**.
2 In the **Settings** window for **Point Probe Expression**, locate the **Expression** section.
3 In the **Expression** text field, type `v_solid`.
4 In the **Variable name** text field, type `v`.

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 `range(0,5e-2,5)`.
4 Click to expand the **Results While Solving** section. Select the **Plot** check box.
5 From the **Update at** list, choose **Time steps taken by solver**.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

Spatial mesh displacement (comp1.spatial.disp)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 1** node, then click **Spatial mesh displacement (comp1.spatial.disp)**.
2 In the **Settings** window for **Field**, locate the **Scaling** section.
3 In the **Scale** text field, type `1e-3`.

Displacement field (comp1.u_solid)

1 In the **Model Builder** window, click **Displacement field (comp1.u_solid)**.
2 In the **Settings** window for **Field**, locate the **Scaling** section.
3 In the **Scale** text field, type `1e-3`.

Pressure (comp1.p)

1 In the **Model Builder** window, click **Pressure (comp1.p)**.
2 In the **Settings** window for **Field**, locate the **Scaling** section.
3 From the **Method** list, choose **Manual**.
4 In the **Scale** text field, type `1e3`.

Velocity field (spatial frame) (comp1.u_fluid)

1 In the **Model Builder** window, click **Velocity field (spatial frame) (comp1.u_fluid)**.
2 In the **Settings** window for **Field**, locate the **Scaling** section.
3 From the **Method** list, choose **Manual**.
4 In the **Scale** text field, type `1`.
5 In the **Study** toolbar, click **Compute**.

RESULTS

Velocity (spf)

The first plot group shows the fluid velocity magnitude.

Surface 2

1 Right-click **Velocity (spf)** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `solid.mises`.
4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Traffic**.

Arrow Surface 1

Right-click **Velocity (spf)** and choose **Arrow Surface**.

Animation 1

In the **Velocity (spf)** toolbar, click **Animation** and choose **Player**.

Probe Plot Group 5

1 In the **Settings** window for **1D Plot Group**, type `Lift and drag forces` in the **Label** text field.
2 Locate the **Plot Settings** section. Select the **x-axis label** check box.
3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
4 In the **Title** text area, type `Lift and drag forces (N)`.
5 Locate the **Legend** section. From the **Position** list, choose **Middle left**.
6 In the **Lift and drag forces** toolbar, click **Plot**.
7 Locate the **Axis** section. Select the **Manual axis limits** check box.
8 In the **x minimum** text field, type `3`.
9 In the **x maximum** text field, type `5`.
10 In the **y minimum** text field, type `-160`.
11 In the **y maximum** text field, type `500`.

12 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

13 In the **Lift and drag forces** toolbar, click **Plot**.

Probe Plot Group 6

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 6**.

2 In the **Settings** window for **1D Plot Group**, type `Beam tip displacement` in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **x-axis label** check box.

4 Locate the **Axis** section. Select the **Manual axis limits** check box.

5 In the **x minimum** text field, type `3`.

6 In the **x maximum** text field, type `5`.

7 In the **y minimum** text field, type `-40`.

8 In the **y maximum** text field, type `40`.

9 Locate the **Title** section. From the **Title type** list, choose **Manual**.

10 In the **Title** text area, type `Beam tip displacement (mm)`.

11 In the **Beam tip displacement** toolbar, click **Plot**.

1D Plot Group 7

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

2 In the **Settings** window for **1D Plot Group**, type `Frequency spectrum` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Domain Point Probe 1**.

4 From the **Time selection** list, choose **Interpolated**.

5 In the **Times (s)** text field, type `range(3,5e-3,5)`.

Point Graph 1

1 Right-click **Frequency spectrum** 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 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Frequency spectrum**.

6 Select the **Frequency range** check box.

7 In the **Minimum** text field, type `1`.

8 In the **Maximum** text field, type `15`.

Point Graph 2

1 Right-click **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 In the **Frequency spectrum** toolbar, click **Plot**.

1D Plot Group 8

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

2 In the **Settings** window for **1D Plot Group**, type `Beam tip trajectory` in the **Label** text field.

Table Graph 1

1 Right-click **Beam tip trajectory** and choose **Table Graph**.

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

3 From the **x-axis data** list, choose **Displacement field, X component (mm), Point: (0.595, 0.2)**.

4 From the **Plot columns** list, choose **Manual**.

5 In the **Columns** list, select **Displacement field, Y component (mm), Point: (0.595, 0.2)**.

6 In the **Beam tip trajectory** toolbar, click **Plot**.

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.

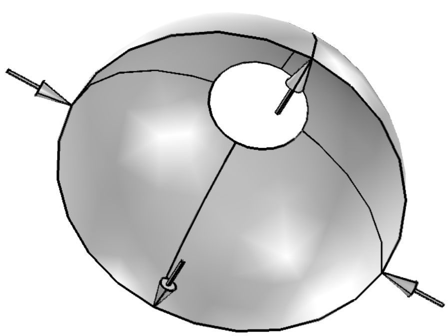


Figure 1: The geometry and loads.

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 equivalent stress.

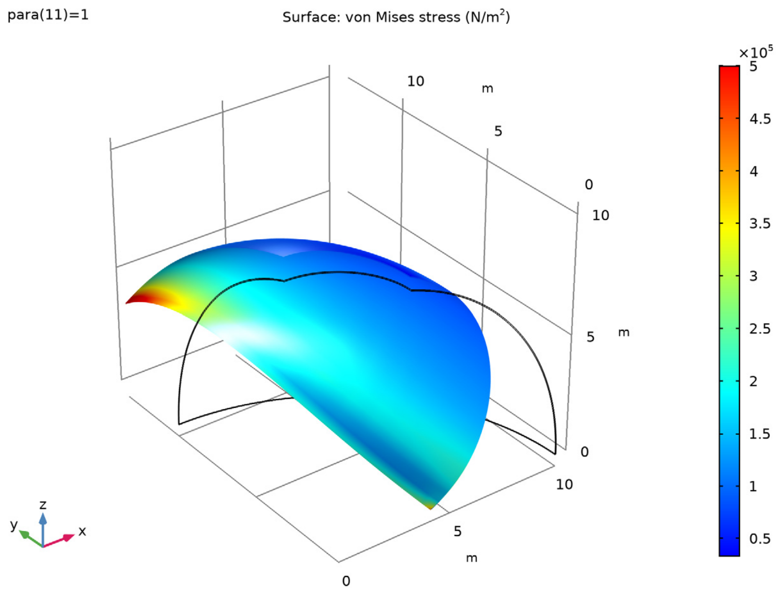


Figure 2: von Mises stress on top surface.

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.

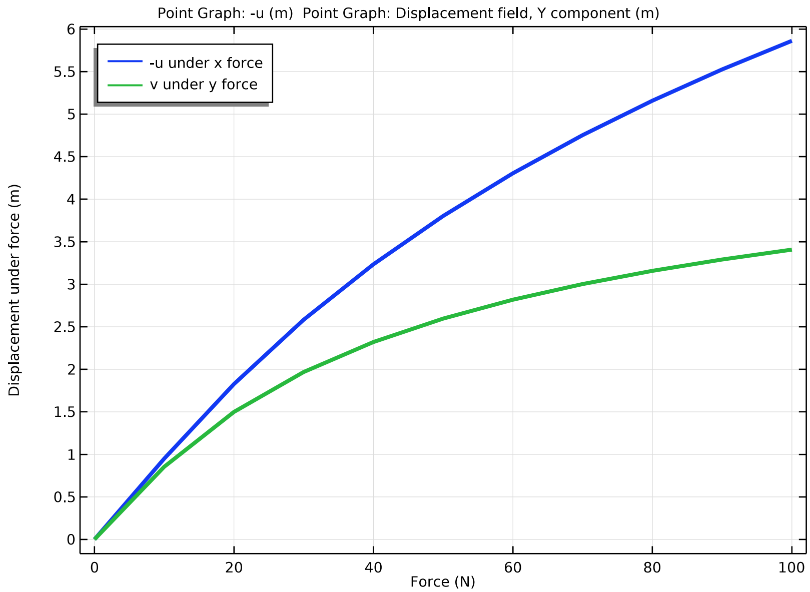


Figure 3: Displacements as functions of applied load.

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 **General Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

Sphere 1 (sph1)

1 In 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 Click **Build Selected**.

Block 1 (blk1)

1 In 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 Click **Build Selected**.

Difference 1 (dif1)

1 In 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 **Activate selection** toggle button.

5 Select the object **blk1** only.

6 Click **Build Selected**.

Convert to Surface 1 (csur1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Surface**.

2 Select the object **dif1** only.

3 In the **Settings** window for **Convert to Surface**, click **Build Selected**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, 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 In the **Settings** window for **Delete Entities**, click **Build Selected**.

4 Click the **Zoom Extents** button in 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 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	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)

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Thickness and Offset 1**.

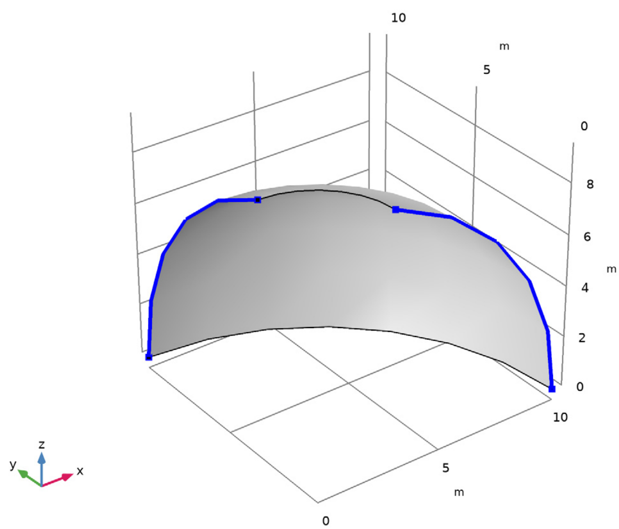
2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

3 In the d text field, type `0.04`.

Symmetry 1

1 In the **Physics** toolbar, click **Edges** and choose **Symmetry**.

2 Select Edges 1 and 4 only.



Prescribed Displacement/Rotation 1

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.

2 Select Point 4 only.

It might be easier to select the correct point by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)

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 In the **Physics** toolbar, click **Points** and choose **Point Load**.

2 In the **Settings** window for **Point Load**, type `Point Load, X` in the **Label** text field.

3 Select Point 4 only.

4 Locate the **Force** section. Specify the $\mathbf{F}_\mathrm{P}$ vector as

-100*para	x
0	y
0	z

Point Load 2

1 In the **Physics** toolbar, click **Points** and choose **Point Load**.

2 In the **Settings** window for **Point Load**, type `Point Load, Y` in the **Label** text field.

3 Select Point 2 only.

4 Locate the **Force** section. 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 **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 type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 16.

6 In the **Element ratio** text field, type 3.

7 From the **Growth formula** list, choose **Geometric sequence**.

Distribution 2

1 In the **Model Builder** window, 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 type** list, choose **Predefined**.

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 **Growth formula** list, choose **Geometric sequence**.

9 Click **Build All**.

GLOBAL DEFINITIONS

Parameters 1

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

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, under **Study 1** 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. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list
para (Solver parameter)	`range(0,0.1,1)`

Solution 1 (sol1)

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

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

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `0.0001`.

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

6 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

7 From the **Nonlinear method** list, choose **Constant (Newton)**.

8 In the **Study** toolbar, click **Compute**.

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Results>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 In the **Stress (shell)** toolbar, click **Plot**.

1D Plot Group 5

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

Point Graph 1

1 Right-click **1D Plot Group 5** and choose **Point Graph**.

2 Select Point 4 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
4 In the **Expression** text field, type `-u`.
5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6 In the **Expression** text field, type `para*100[N]`.
7 Click to expand the **Coloring and Style** section. In the **Width** text field, type `3`.
8 Click to expand the **Legends** section. Select the **Show legends** check box.
9 From the **Legends** list, choose **Manual**.
10 In the table, enter the following settings:

Legends
-u under x force

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.
2 In the **Settings** window for **Point Graph**, locate the **Selection** section.
3 Select the **Activate selection** 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 5

1 In the **Model Builder** window, click **1D Plot Group 5**.
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 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

8 In the **1D Plot Group 5** toolbar, click **Plot**.

Evaluate the displacements in the points where a comparison should be made with the target.

Evaluation Group 1

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, locate the **Data** section.

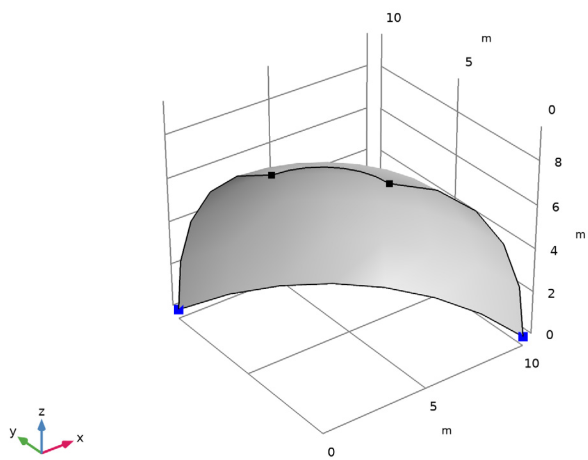
3 From the **Parameter selection (para)** list, choose **Last**.

4 Locate the **Transformation** section. Select the **Transpose** check box.

Point Evaluation 1

1 Right-click **Evaluation Group 1** and choose **Point Evaluation**.

2 Select Points 2 and 4 only.



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

4 In the table, enter the following settings:

Expression	Unit	Description
`u`	`m`	`Displacement field, X component`
`v`	`m`	`Displacement field, Y component`

5 In the **Evaluation Group 1** toolbar, click **Evaluate**.

Created in COMSOL Multiphysics 5.5

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).

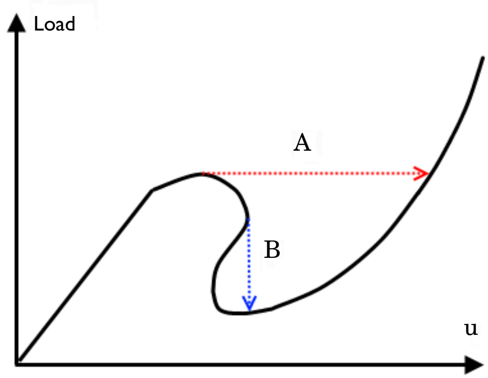


Figure 1: Load versus displacement in snap-through buckling

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.

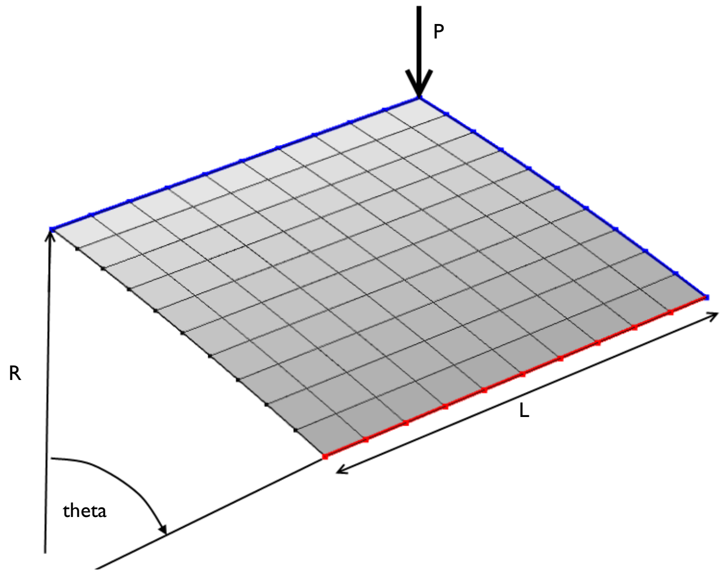


Figure 2: Problem description.

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).

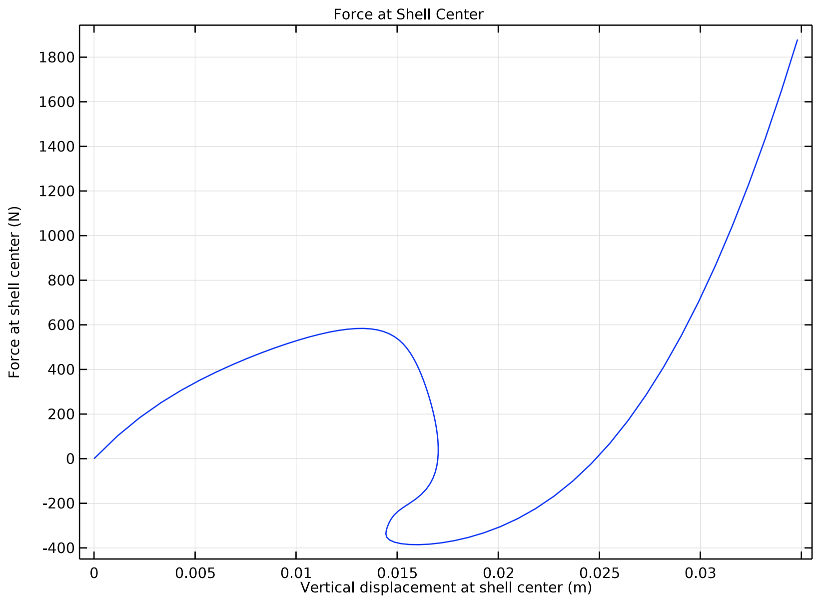


Figure 3: Applied load versus panel center displacement.

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 monotonically.

In this example, a good such parameter is the average of the displacement in the direction of the applied force. You use a nonlocal average coupling 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 **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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
thic	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 In 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**.

Work Plane 1 (wp1)>Line Segment 1 (ls1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Line Segment**.

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

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **yw** text field, type `R`.

6 Locate the **Endpoint** section. In the **xw** text field, type `L` and **yw** to `R`.

7 Click **Build Selected**.

8 In the **Model Builder** window, click **Geometry 1**.

Revolve 1 (rev1)

1 In 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 Click **Build Selected**.

DEFINITIONS

Click the **Zoom Extents** button in the **Graphics** toolbar.

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal 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 In the **Definitions** toolbar, click **Nonlocal 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 In 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)

Thickness and Offset 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Thickness and Offset 1**.
2. In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.
3. In the d text field, type `thic`.

Linear Elastic Material 1

1. In the **Model Builder** window, 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 **Physics** toolbar, click **Edges** and choose **Symmetry**.
2. Select Edge 3 only.

Symmetry 2

1. In the **Physics** toolbar, click **Edges** and choose **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. In the **Physics** toolbar, click **Edges** and choose **Pinned**.
2. Select Edge 2 only.

Point Load 1

1. In the **Physics** toolbar, click **Points** and choose **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 Click the **Show More Options** button in the **Model Builder** toolbar.

6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

7 Click **OK**.

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **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. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `force` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Force (N)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `displacement` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Displacement (m)**.

14 Click **OK**.

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 **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, under **Study 1** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.
3 Select the **Auxiliary sweep** check box.
4 Click **Add**.
5 In the table, enter the following settings:

Parameter name	Parameter value list
disp (Displacement parameter)	`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 In 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 (>=1)	√	`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, 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

1D Plot Group 5

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

2 In the **Settings** window for **1D Plot Group**, type `Force at Shell Center` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Force at Shell Center`.

Point Graph 1

1 Right-click **Force at Shell Center** and choose **Point Graph**.

2 Select Point 4 only.

3 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 - N**.

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

5 In the **Expression** text field, type `w_center`.

6 Select the **Description** check box.

7 In the associated text field, type `Vertical displacement at shell center`.

8 In the **Force at Shell Center** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.5

Random Vibration Analysis of a Deep Beam

Introduction

This example studies forced random vibrations of a simply-supported deep beam. The beam is loaded by a distributed force with a uniform power spectral density (PSD). The output PSD are computed for the displacement and bending stress response. The computed values are compared with analytical results (NAFEMS test 5R from Ref. 1).

Model Definition

The model studied in this example consists of a simply supported beam with a square cross section. One end is pinned and has a constrained rotation along the beam axis. At the other end, the displacements in the plane of beam cross section are constrained.

GEOMETRY

- Beam length, L = 10 m
- Beam cross section dimension l = 2 m

With such aspect ratio of the cross section size to the beam length, shear deformations and rotational inertia effects can no longer be neglected as it is done in the Euler-Bernoulli theory. Therefore, the solution is computed using a Timoshenko beam.

MATERIAL

- Young's modulus, E = 200 GPa
- Poisson's ratio, $\nu = 0.3$
- Mass density, $\rho = 8000\ \mathrm{kg/m^3}$
- Rayleigh damping coefficient: $\alpha = 5.36\ \mathrm{s^{-1}}$, $\beta = 7.46\cdot 10^{-5}$ m/s

The values of the damping coefficients are chosen to give a damping ratio of 2% for the first eigenmode.

CONSTRAINTS

At x = 0, $u = v = w = 0$; $thx = 0$

At x = 10, $v = w = 0$

LOAD

For a linear system, the response in the frequency domain for a single variable V to the excitation F can be written

$$V(f) = H_{VF}(f)F$$

where f is the frequency, and H is the complex valued transfer function. It can then be shown that the corresponding spectral densities have the relation

$$S_V(f) = \left|H_{VF}(f)\right|^2 S_F(f) = H^*{}_{VF}(f) H_{VF}(f) S_F(f)$$

where the asterisk denotes a complex conjugate. This type of relation is true not only for the degrees of freedom, but for any quantity that is linearly related to the input. This includes components of stress and (engineering) strain, but not nonlinear quantities such as equivalent or principal stresses.

In this example, a load of $F = 10^6$ N/m in the y direction is applied uniformly along the beam for the forced harmonic vibration study. For the random vibration analysis, the load is assumed to have a uniformly distributed PSD of 10^{12} $(\mathrm{N/m})^2/\mathrm{Hz}$. Thus, one should expect that results have the property

$$S_V(f) = |V|^2(f)$$

That is, the PSD response is simply the square of the standard harmonic response.

The plot below shows the computed PSD of the beam vertical displacement at the mid point. Note that is also matches the squared non-random frequency response at the same point.

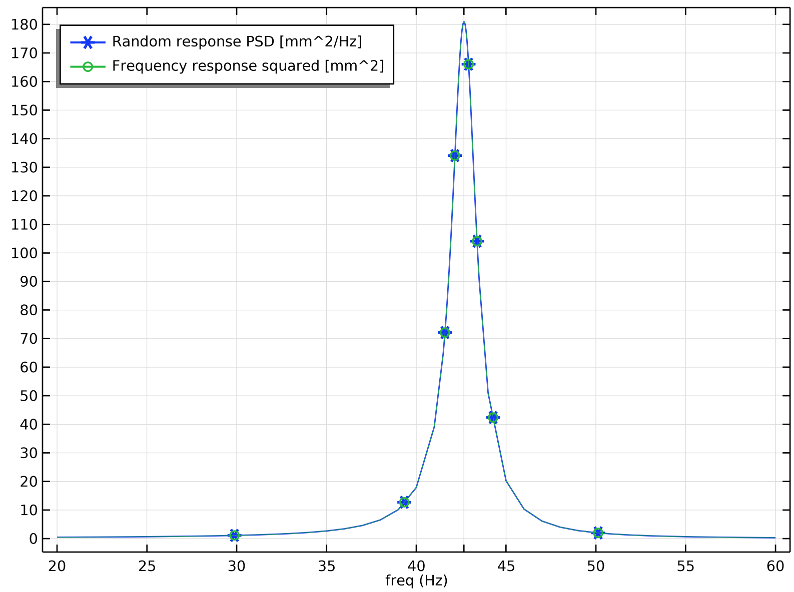


Figure 1: The PSD of the displacement response at the midpoint of the beam.

In Table 1, the computed results are compared with the analytical results from Ref. 1. The agreement is good.

TABLE 1: COMPARISON BETWEEN ANALYTICAL AND COMPUTED RANDOM RESPONSES.

	Peak displacement PSD mm^2/Hz	**Peak stress PSD** $(N/mm^2)^2/Hz$	**Frequency** Hz
REFERENCE	180.90	58516	42.65
COMSOL	180.89	56933	42.66

In this benchmark, a mesh consisting of only five elements is prescribed. The stress is measured at the midpoint of the beam, that is at the midpoint of the central beam element. Since the finite element approximation in the beam elements give a linear variation of the bending moment within each element, the bending moment (and thus the stress) in the central element is constant for symmetry reasons. The true midpoint value will thus be underestimated. If six elements are used instead, there will be a node at the midpoint. The stress PSD value in that node turns out to be 60,652 $(\mathrm{N/mm^2})^2/\mathrm{Hz}$.

Reference

1. J. Maguire, D.J. Dawswell, and L. Gould, "Selected Benchmarks for Forced Vibration", *NAFEMS R0016*, 1989.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/random_vibration_deep_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 for Selected Physics Interfaces> Random Vibration (PSD)**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
F	1e6[N/m]	1E6 N/m	Edge load
PSD	F^2/1[Hz]	1E12 kg^2/s^3	Random edge load, power spectral density

GEOMETRY 1

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

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

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 In the **x** text field, type 10.

6 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	Variable	Value	Unit	Property group
Young's modulus	E	2e11	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	8000	kg/m^3	Basic

BEAM (BEAM)

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

2 In the **Settings** window for **Beam**, locate the **Beam Formulation** section.

3 From the list, choose **Timoshenko**.

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 `2`.

5 In the h_z text field, type `2`.

Section Orientation 1

1 In the **Model Builder** window, 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

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Damping 1

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.

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

3 In the α_{dM} text field, type `5.36`.

4 In the β_{dK} text field, type `7.46e-5`.

Prescribed Displacement/Rotation 1

1 In 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. From the list, choose **Rotation**.

8 Select the **Free rotation around y direction** check box.

9 Select the **Free rotation around z direction** check box.

Prescribed Displacement/Rotation 2

1 In 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.

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 Edge 1 only.

Distribution 1

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

2 In the **Settings** window for **Edge**, click **Build All**.

DEFINITIONS

Set up an operator to evaluate variables at the beam midpoint.

General Extrusion 1 (genext1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **General Extrusion**.

2 In the **Settings** window for **General Extrusion**, locate the **Source Selection** section.

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

4 Select Edge 1 only.

5 Locate the **Destination Map** section. In the **x-expression** text field, type `5`.

6 In the **y-expression** text field, type `0`.

7 In the **z-expression** text field, type `0`.

8 Locate the **Source** section. From the **Source frame** list, choose **Material (X, Y, Z)**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
V	genext1(v)	m	Displacement, y component
Sb	genext1(beam.sb1)	N/m²	Bending stress

GLOBAL DEFINITIONS

Reduced-Order Modeling

Set up a control parameter to be used as the edge load.

Global Reduced Model Inputs 1

1 In the **Model Builder** window, expand the **Global Definitions>Reduced-Order Modeling** node, then click **Global Reduced Model Inputs 1**.

2 In the **Settings** window for **Global Reduced Model Inputs**, locate the **Reduced Model Inputs** section.

3 In the table, enter the following settings:

Control name	Expression
Fy	F

BEAM (BEAM)

Edge Load 1

1 In the **Physics** toolbar, click **Edges** and choose **Edge Load**.

2 Select Edge 1 only.

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

4 Specify the $\mathbf{F}_L$ vector as

0	X
Fy	Y
0	Z

STUDY 1

Step 1: Eigenfrequency

Set the search position close to the target value of the first natural frequency.

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 In the **Search for eigenfrequencies around** text field, type 40.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Beam (beam)> Linear Elastic Material 1>Damping 1**.

6 Click **Disable**.

The eigenmode computation should be always performed for the undamped system. The damping will be used however in the consequent modal frequency response and random response analysis.

STUDY 3

Model Reduction

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

2 In the **Settings** window for **Model Reduction**, locate the **Model Reduction Settings** section.

3 Select the **Reduced model simulation** check box.

The computation of the solution for Study 3 will find the eigenfrequencies and build up a modal reduced-order model (ROM) based on the computed eigenmodes.

4 In the **Home** toolbar, click **Compute**.

You can see all computed eigenfrequencies in the automatically generated evaluation group.

GLOBAL DEFINITIONS

Next, set up the input PSD for the random edge load.

Reduced-Order Modeling

1 In the **Model Builder** window, under **Global Definitions>Reduced-Order Modeling** click **Random Vibration 1 (rvib1)**.

2 In the **Settings** window for **Random Vibration**, locate the **Power Spectrum** section.

3 In the table, enter the following settings:

Control name	Power spectral density
rom1.Fy	PSD

Update the study to make the input change available for the solution.

STUDY 3

1 In the **Study** toolbar, click **Update Solution**.

The random response computations can be performed as postprocessing steps using the updated solution.

RESULTS

Add a plot of the PSD for the displacement responses at the midpoint. For verification, you also plot the non-random frequency response result computed using ROM.

Global Evaluation Sweep 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Other>Global Evaluation Sweep**.

Use the frequency range to resolve well the values close to the target first natural frequency.

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

3 In the table, enter the following settings:

Parameter name	Parameter value list
freq	range(20,1,41) range(41.5,0.01,43.5) range(44,1,60) [Hz]

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

Expression	Unit	Description
rvib1.psd(V)*1e6		Random response PSD [mm^2/Hz]
abs(rom1.eval(V))^2*1e6		Frequency response squared [mm^2]

Tables

Click **Evaluate**.

TABLE

1 Go to the **Table** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Table Graph 1

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Results>1D Plot Group 3>Table Graph 1**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

4 Locate the **Legends** section. Select the **Show legends** check box.

1D Plot Group 3

Indicate the target peak frequency.

1 In the **Model Builder** window, click **1D Plot Group 3**.

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

3 In the **Extra x** text field, type `42.65`.

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

5 In the **1D Plot Group 3** toolbar, click **Plot**.

The actually computed peak frequency is close to 42.66 (Hz).

Finally, calculate the maximum PSD values in the computed frequency range for both the displacement and bending stress responses.

Global Evaluation Sweep 2

1 In the **Model Builder** window, under **Results>Derived Values** right-click **Global Evaluation Sweep 1** and choose **Duplicate**.

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

3 In the table, click to select the cell at row number 1 and column number 2.

4 In the table, enter the following settings:

Parameter name	Parameter value list
freq	42.66 [Hz]

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

Expression	Unit	Description
rvib1.psd(V)*1e6		Displacement, y component, maximum PSD [mm^2/Hz]
rvib1.psd(Sb)/1e12		Bending stress, maximum PSD [(N/mm^2)^2/Hz]

6 Click **New Table**.

Compare the results with the target values.

Created in COMSOL Multiphysics 5.5

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.

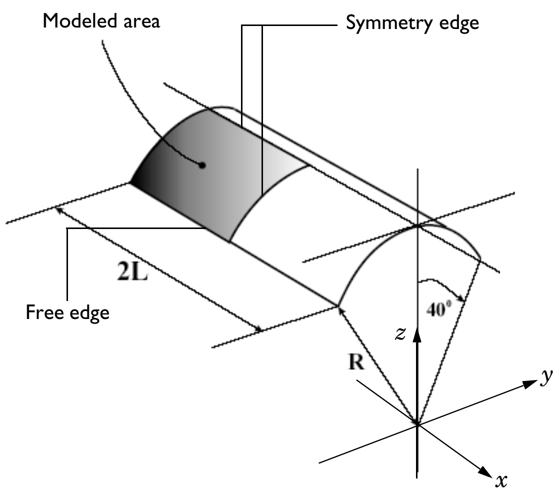


Figure 1: The Scordelis-Lo roof shell benchmark geometry.

- 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 $-90\ \mathrm{N/m^2}$ 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.

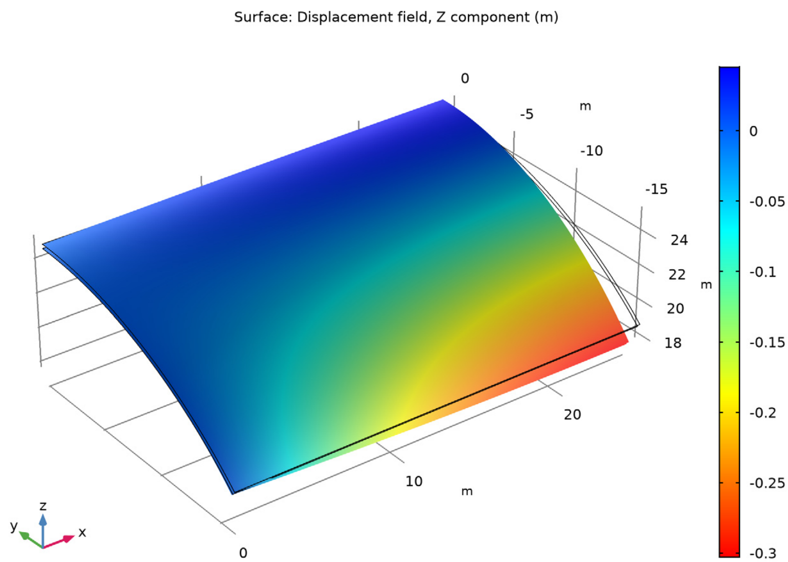


Figure 2: z-displacement with 176 triangular elements.

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.

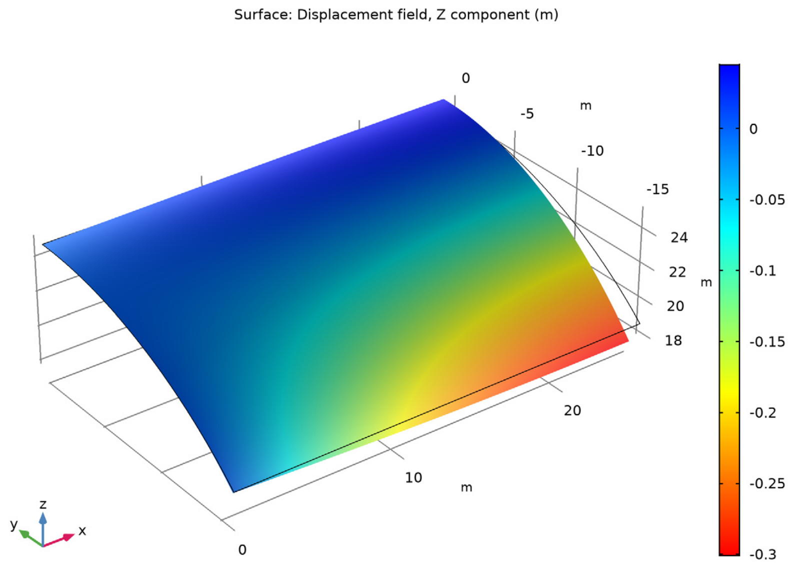


Figure 3: z-displacement with 70 quadrilateral elements.

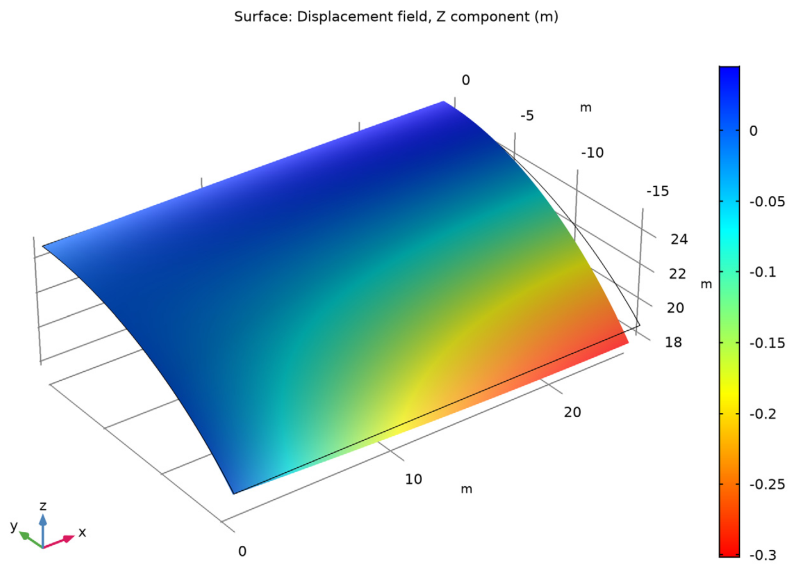


Figure 4: z-displacement with 580 quadrilateral elements.

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 **General Studies>Stationary**.
6. Click **Done**.

GEOMETRY 1

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

Right-click **Work Plane 1 (wp1)** and choose **Show Work Plane**.

Work Plane 1 (wp1)>Polygon 1 (pol1)

1. In the **Work Plane** toolbar, click **Polygon**.
2. In the **Settings** window for **Polygon**, locate the **Coordinates** section.
3. In the table, enter the following settings:

xw (m)	yw (m)
0	25
25	25

4. Right-click **Polygon 1 (pol1)** and choose **Build All Objects**.

Work Plane 1 (wp1)

In the **Model Builder** window, click **Work Plane 1 (wp1)**.

Revolve 1 (rev1)

1 In 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 Click **Build Selected**.
9 Click the **Zoom Extents** button in the **Graphics** toolbar.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.
2 Click **Build Selected**.

SHELL (SHELL)

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Thickness and Offset 1**.
2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.
3 In the d text field, type `0.25`.

Symmetry 1

1 In the **Physics** toolbar, click **Edges** and choose **Symmetry**.
2 Select Edges 3 and 4 only.

Prescribed Displacement/Rotation 1

1 In the **Physics** toolbar, click **Edges** 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 In the **Physics** toolbar, click **Boundaries** and choose **Face Load**.

2 Select Boundary 1 only.

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	Variable	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

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

2 In the **Settings** window for **Study**, type `Study 1: Tri Normal` in the **Label** text field.

3 In the **Home** toolbar, click **Compute**.

RESULTS

Stress (shell)

1 In the **Settings** window for **3D Plot Group**, type `Vertical displacement` in the **Label** text field.

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

Surface 1

1 In the **Model Builder** window, expand the **Results>Vertical displacement** node, then click **Surface 1**.

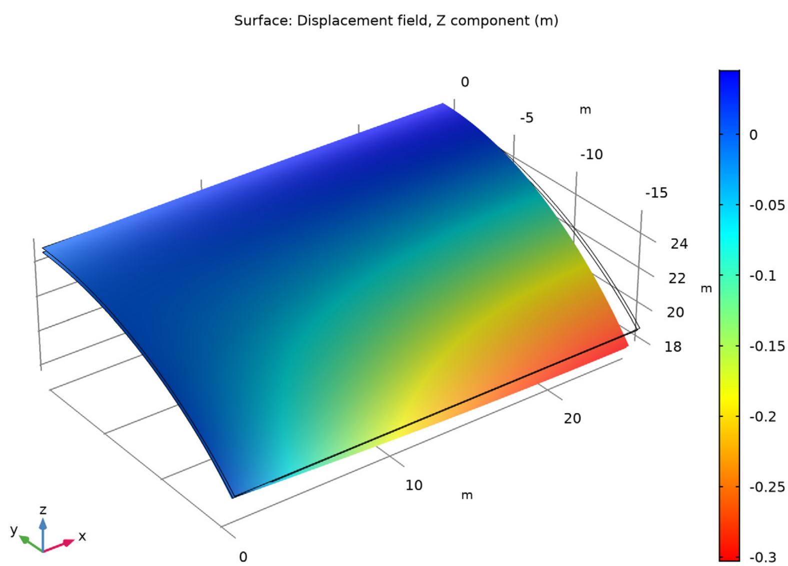
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Shell>Displacement> Displacement field - m>w - Displacement field, Z component**.

3 Locate the **Coloring and Style** section. Select the **Reverse color table** check box.

Vertical displacement

1 In the **Model Builder** window, click **Vertical displacement**.

2 In the **Vertical displacement** toolbar, click **Plot**.



Study 1: Tri Normal/Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1: Tri Normal/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.

MESH 2

1 In the **Mesh** toolbar, click **Add Mesh**.

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

Mapped 1

1 Right-click **Quad Normal** and choose **More Operations>Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.

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

4 Click **Build All**.

ADD STUDY

1 In 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 **General Studies>Stationary**.

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

5 In 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**, type `Study 2: Quad Normal` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

4 In the **Home** toolbar, click **Compute**.

RESULTS

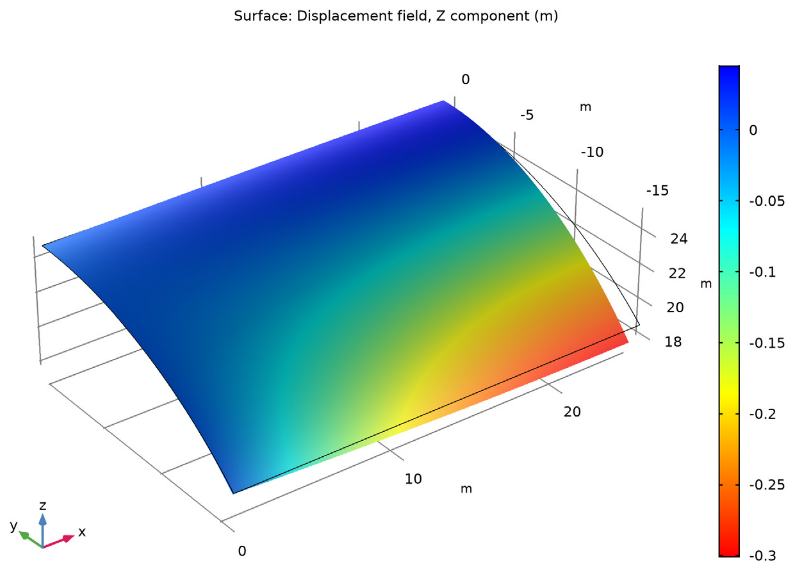
Vertical displacement

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 **Dataset** list, choose **Study 2: Quad Normal/Solution 2 (sol2)**.

4 In the **Vertical displacement** toolbar, click **Plot**.



Study 2: Quad Normal/Solution 2 (sol2)

1 In the **Model Builder** window, under **Results>Datasets** click **Study 2: Quad Normal/ 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.

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 In 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 **General Studies>Stationary**.

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

5 In 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**, type `Study 3: Quad Extra finel` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

4 In the **Home** toolbar, click **Compute**.

RESULTS

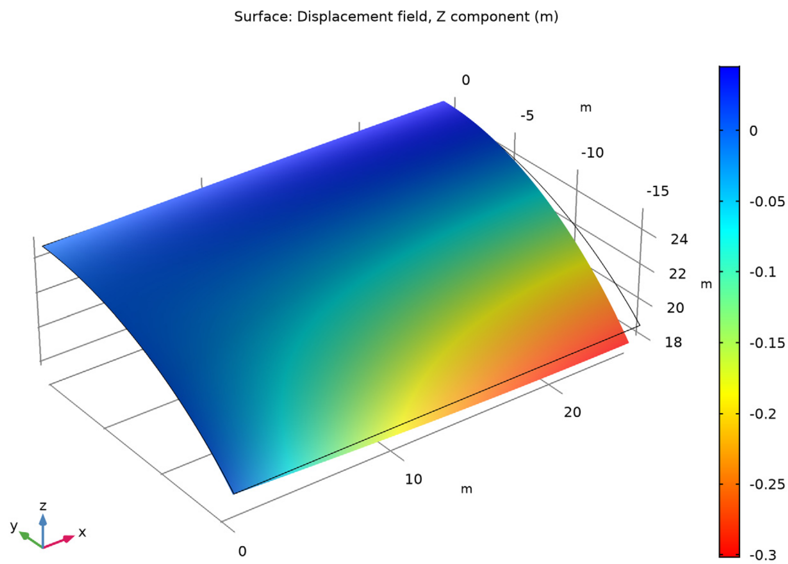
Vertical displacement

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 **Dataset** list, choose **Study 3: Quad Extra finel/Solution 3 (sol3)**.

4 In the **Vertical displacement** toolbar, click **Plot**.



Study 3: Quad Extra fine/Solution 3 (sol3)

1 In the **Model Builder** window, under **Results>Datasets** click **Study 3: Quad Extra fine/ 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.

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 In 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 **General Studies>Stationary**.

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

5 In 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**, type `Study 4: Tri Extra fine` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

4 In the **Home** toolbar, click **Compute**.

RESULTS

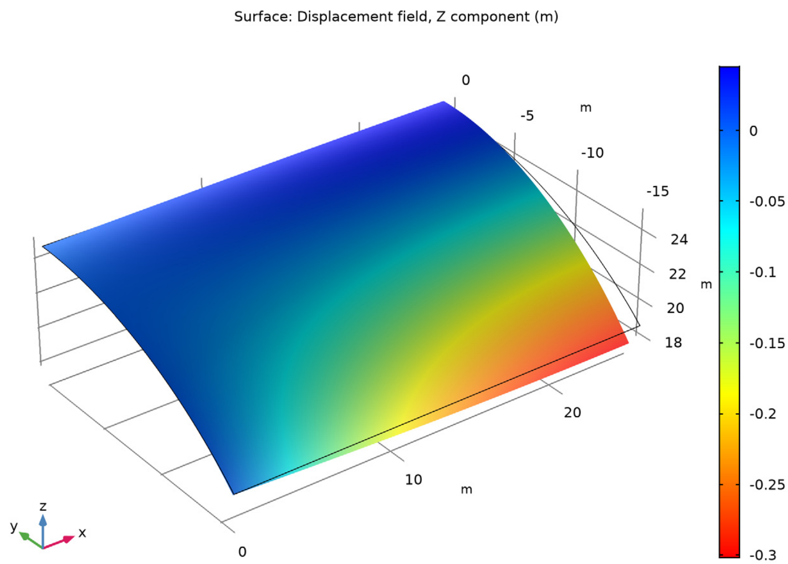
Vertical displacement

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 **Dataset** list, choose **Study 4: Tri Extra fine/Solution 4 (sol4)**.

4 In the **Vertical displacement** toolbar, click **Plot**.



Study 4: Tri Extra fine/Solution 4 (sol4)

1 In the **Model Builder** window, under **Results>Datasets** click **Study 4: Tri Extra fine/ 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.

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 In 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 **General Studies>Stationary**.

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

5 In 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**, type `Study 5: Tri Coarser` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

4 In the **Home** toolbar, click **Compute**.

RESULTS

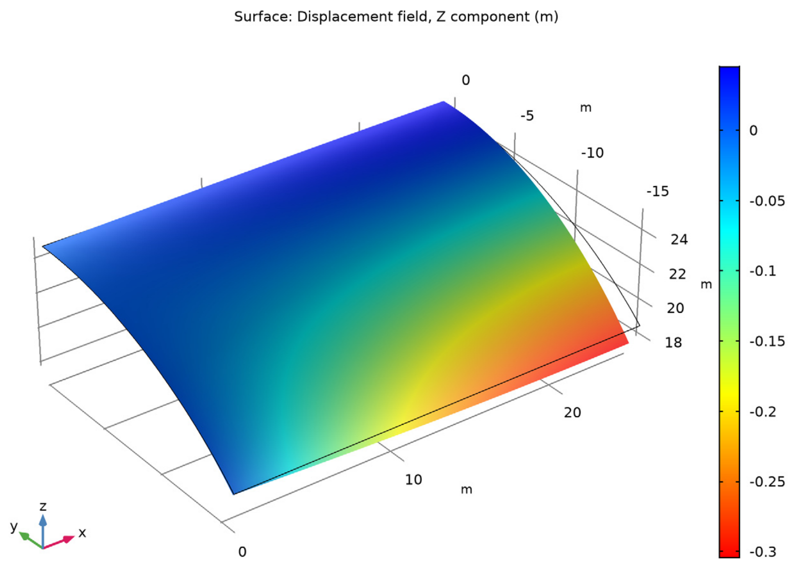
Vertical displacement

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 **Dataset** list, choose **Study 5: Tri Coarser/Solution 5 (sol5)**.

4 In the **Vertical displacement** toolbar, click **Plot**.



Study 5: Tri Coarser/Solution 5 (sol5)

1 In the **Model Builder** window, under **Results>Datasets** click **Study 5: Tri Coarser/ 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.

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 In 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 **General Studies>Stationary**.

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

5 In 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**, type `Study 6: Quad Coarser` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

4 In the **Home** toolbar, click **Compute**.

RESULTS

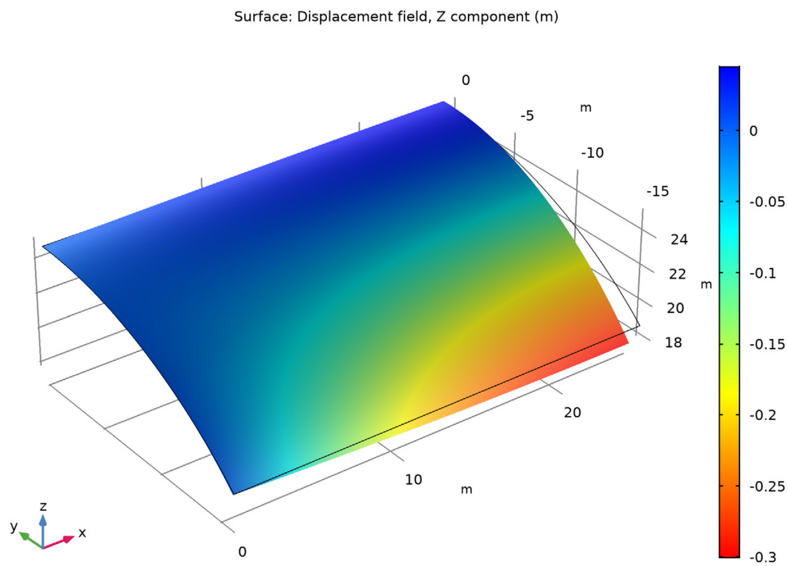
Vertical displacement

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 **Dataset** list, choose **Study 6: Quad Coarser/Solution 6 (sol6)**.

4 In the **Vertical displacement** toolbar, click **Plot**.



Study 6: Quad Coarser/Solution 6 (sol6)

1 In the **Model Builder** window, under **Results>Datasets** click **Study 6: Quad Coarser/ Solution 6 (sol6)**.

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

The following section compares the maximum deformation of midpoint in vertical direction for different element types and mesh densities.

Point Evaluation 1

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

2 Select Point 3 only.

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

4 In the table, enter the following settings:

Expression	Unit	Description
`w`	`m`	`Midpoint displacement, Tri Normal`

5 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 **Dataset** list, choose **Quad Normal (sol2)**.

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

Expression	Unit	Description
`w`	`m`	`Midpoint displacement, Quad Normal`

5 Click **Table 1 - Point Evaluation 1**.

Point Evaluation 3

1 Right-click **Point Evaluation 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Quad Extra fine (sol3)**.

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

Expression	Unit	Description
`w`	`m`	`Midpoint displacement, Quad Extra fine`

5 Click **Table 1 - Point Evaluation 1**.

Point Evaluation 4

1 Right-click **Point Evaluation 3** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Tri Extra fine (sol4)**.

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

Expression	Unit	Description
`w`	`m`	`Midpoint displacement, Tri Extra fine`

5 Click **Table 1 - Point Evaluation 1**.

Point Evaluation 5

1 Right-click **Point Evaluation 4** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Tri Coarser (sol5)**.

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

Expression	**Unit**	**Description**
`w`	`m`	`Midpoint displacement, Tri Coarser`

5 Click **Table 1 - Point Evaluation 1**.

Point Evaluation 6

1 Right-click **Point Evaluation 5** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Quad Coarser (sol6)**.

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

Expression	**Unit**	**Description**
`w`	`m`	`Midpoint displacement, Quad Coarser`

5 Click **Table 1 - Point Evaluation 1**.

Created in COMSOL Multiphysics 5.5

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.

Determining the stress intensity factor directly from the local state at the crack tip is often problematic, since the stresses are singular there. Because of this, more indirect energy based methods are attractive. In this example, K_I is computed using the J-integral and from the energy release rate.

In addition, the crack growth rate and number of cycles needed to propagate the crack a certain distance are computed.

Model Definition

A plate with a width of 1.5 m and height of 4.5 m has a single horizontal edge-crack of length $a = 0.6$ m at the middle of the left vertical edge, see Figure 1. An external load is pulling the plate such that the top and bottom edges experience tensile stress, σ, of 20 MPa.

The analysis is made using a number of crack lengths ranging from 0.5 m to 0.7 m, so that the influence of the crack length can be studied.

Because of the symmetry, only half of the plate is modeled. Additional domains are created in the half plate rectangle to create a path for integration contours for the J-integral. There are three paths for computing the J-integral:

1. The external boundaries, excluding the crack surface.
2. A path with three straight lines, formed by adding an extra rectangle.
3. A semicircular path, formed by adding a circle to the geometry.

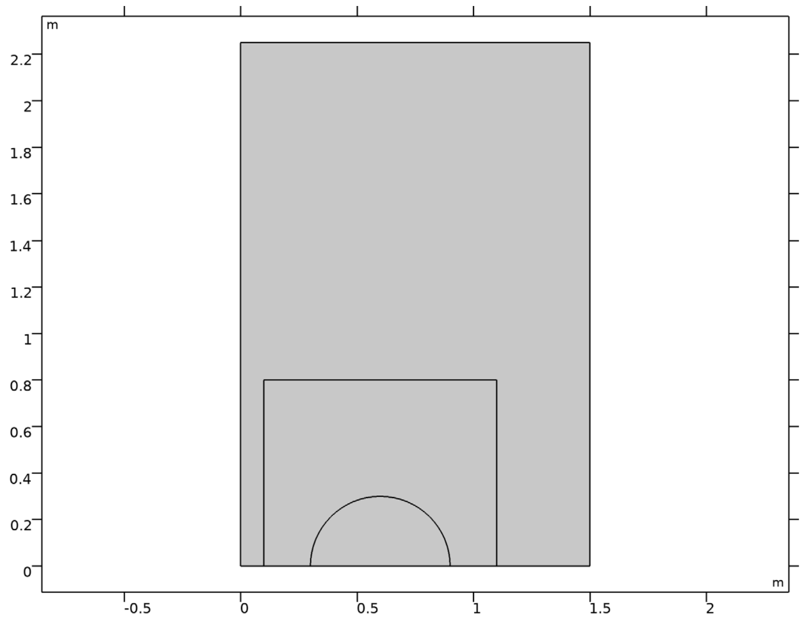


Figure 1: Plate geometry.

You apply a tensile load to the upper horizontal edge, while the lower horizontal edge is constrained in the *y* direction from the crack tip to the right vertical boundary using a symmetry condition. One point is constrained in the horizontal direction in order to suppress rigid body motions.

MATERIAL MODEL

The same material properties are representative for steel.

TABLE 1: MATERIAL DATA.

QUANTITY	NAME	EXPRESSION
Young's modulus	E	$206 \cdot 10^9$ Pa
Poisson's ratio	ν	0.3
Coefficient in Paris' law	A	$1.4 \cdot 10^{-11}$ (K_I unit system: $MN/m^{3/2}$)
Exponent in Paris' law	m	3.1

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 path independent 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 **T** is the traction vector defined as

$$\mathbf{T} = \begin{bmatrix} \sigma_x \cdot n_x + \sigma_{xy} \cdot n_y \\ \sigma_{xy} \cdot n_x + \sigma_y \cdot n_y \end{bmatrix}$$

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.

ENERGY RELEASE RATE

For a linear elastic material it is actually possible to compute the value of the J-integral without using the path integrals. The reason is that its value equals the value of the energy release rate, G,

$$G = -\frac{1}{t}\frac{\partial U}{\partial a} \tag{2}$$

Here U is the potential energy, a is the crack length, and t is the thickness. By computing the potential energy for two slightly different crack lengths, G can be estimated as

$$G = -\frac{1}{t}\frac{\Delta U}{\Delta a} \tag{3}$$

The potential energy of an elastic body is

$$U = \frac{1}{2}\int_{\Omega} \sigma:\varepsilon \, dV - \int_{\partial\Omega} \mathbf{T}\cdot\mathbf{u} \, dS$$

The first term is the strain energy in the volume, and the second term is the potential of the prescribed tractions on the boundary. Because of the linearity,

$$\int_{\Omega} \sigma:\varepsilon \, dV = \int_{\partial\Omega} \mathbf{T}\cdot\mathbf{u} \, dS$$

Thus, it is possible to compute the potential energy using either of these terms independently.

$$U = -\frac{1}{2}\int_{\Omega} \sigma:\varepsilon \, dV = -\frac{1}{2}\int_{\partial\Omega} \mathbf{T}\cdot\mathbf{u} \, dS$$

The total strain energy density exists as a built-in variable, making the first expression attractive for determining G.

CRACK PROPAGATION

When subjected to a periodic load, the crack growth rate (in meters per load cycle) can be expressed by Paris' law:

$$\frac{da}{dN} = A(\Delta K_I)^m \tag{4}$$

Here A and m are material parameters and ΔK_{I} is the range of the stress intensity factor. It is assumed that the load varies between zero and 20 MPa, so that ΔK_{I} equals the computed K_{I}.

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 $K_{\mathrm{Ia}} = 57.7\ \mathrm{MN/m^{3/2}}$.

The calculated stress intensity factors for the three different contours are

CONTOUR	STRESS INTENSITY FACTOR
1	57.8 $MPa \cdot m^{1/2}$
2	57.7 $MPa \cdot m^{1/2}$
3	57.7 $MPa \cdot m^{1/2}$

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.

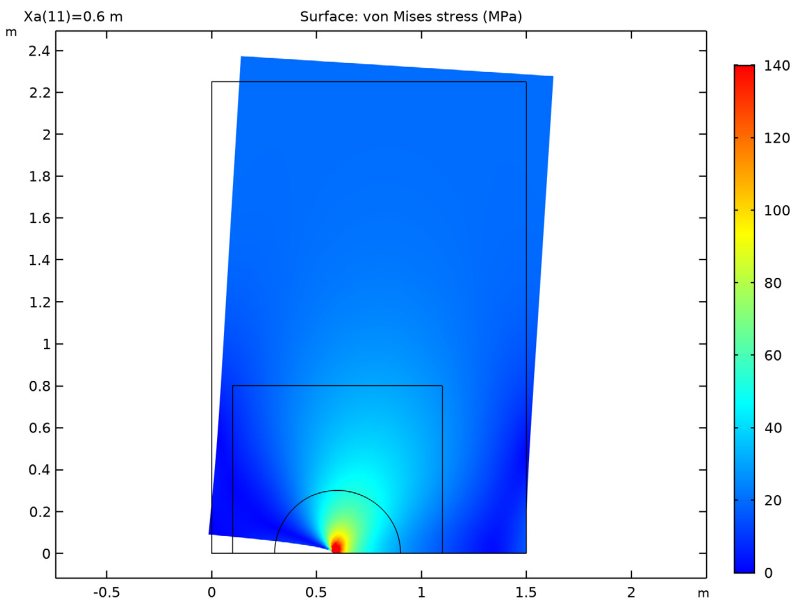


Figure 2: von Mises stresses and the deformed shape of the plate when the crack length is 0.6 m. The displacement is exaggerated to illustrate the deformation under the applied load.

The three different ways of computing the energy release rate, and thus K_I, are compared in Figure 3. As can be seen, all three methods give essentially the same values. You can use the most convenient approach when you need to compute a stress intensity factor.

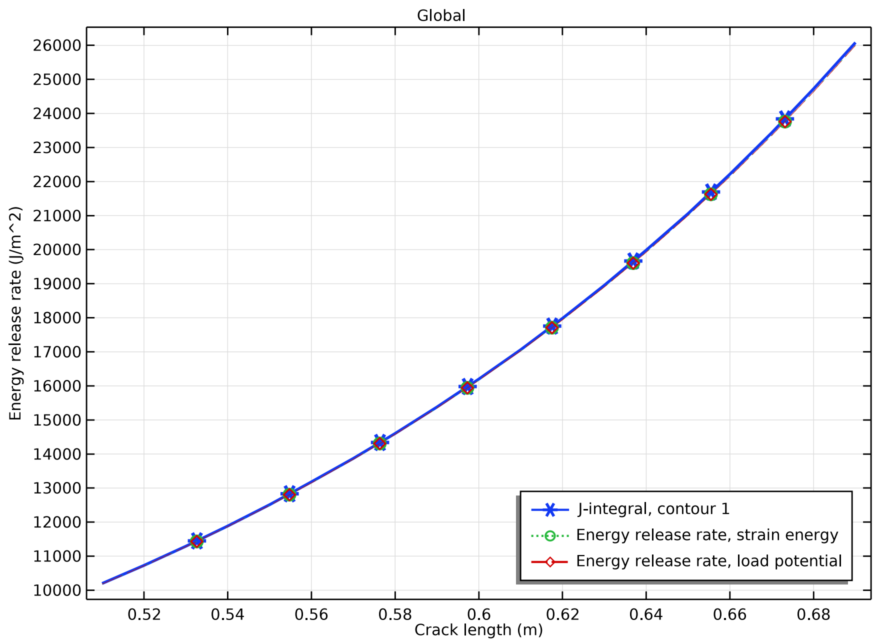


Figure 3: J-integral compared with energy release rates computed using numerical differentiation.

Finally, the crack growth speed can be investigated. In Figure 4, the crack growth speed is shown as function of the crack length. The dependence is quite strong: an increase in crack length from 0.5 m to 0.7 m (40%) increases the crack growth rate by a factor of 5. According to the constants used in Paris' law, the crack growth rate is proportional to the stress intensity factor raised to the power of 3.1. As can be seen from the previous results, the stress intensity factor increases strongly with the crack length, and this combination results in the increase in crack growth rate.

In practice, Paris' law may not be applicable when K_{I} approaches the critical value K_{Ic}.

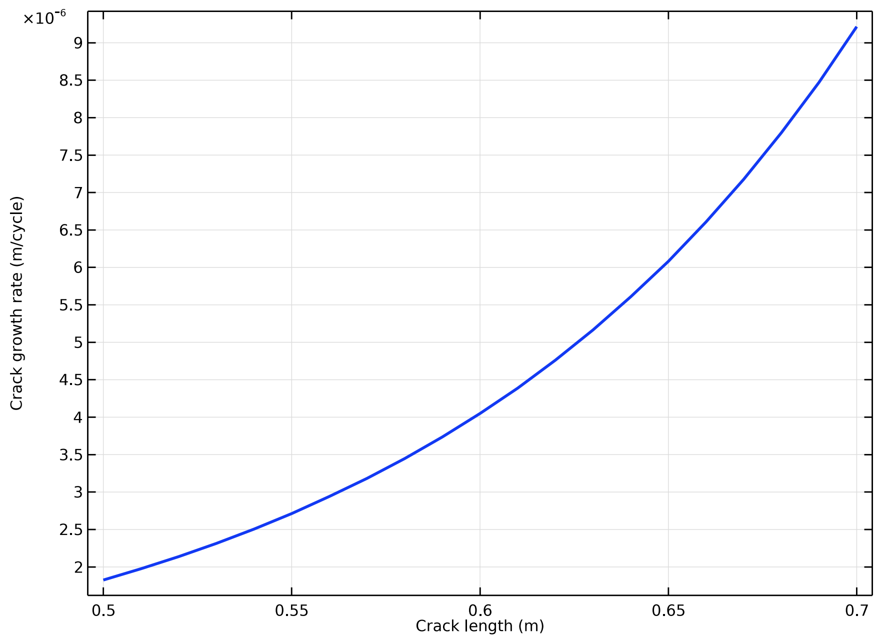


Figure 4: Crack propagation rate as function of the crack length.

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. 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. 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, the built-in normal vector is replaced by a local variable which is reversed when needed. The criterion is based on the sign of the scalar product between the normal to the contour, $\mathbf{n}$, and the vector from the crack tip to the current point on the contour, **r**. If **n** is oriented inward, then this scalar product is negative, and the normal used in the J-integral evaluation must be reversed.

When computing the energy release rates, the derivative of the potential energy is computed using a difference approximation. In order to access different solutions in a single expression, the `withsol()` operator is used.

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 **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `single_edge_crack_parameters.txt`.

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type `Wp`.

4 In the **Height** text field, type `Hp`.

Add the integration paths, one rectangular and one circular.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **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 `max(Xa-0.5[m],0.05[m])`.

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

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

3 In the **Radius** text field, type `0.3`.

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

5 Locate the **Position** section. In the **x** text field, type `Xa`.

6 Click **Build All Objects**.

Add a point at the crack tip.

Point 1 (pt1)

1 In the **Geometry** toolbar, click **Point**.

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

3 In the **x** text field, type `Xa`.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **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	Variable	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 `Th`.

Symmetry 1

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

2 Select Boundaries 8, 9, and 11 only.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** 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
Q0	y

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.

Suppress rigid body motion.

2 Select Point 11 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

Add integration operators for the path integrals.

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `J-integral path 1` in the **Label** text field.

3 In the **Operator name** text field, type `Jpath1`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 1, 3, and 12 only.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `J-integral path 2` in the **Label** text field.

3 In the **Operator name** text field, type `Jpath2`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 4, 6, and 10 only.

Integration 3 (intop3)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `J-integral path 3` in the **Label** text field.

3 In the **Operator name** text field, type `Jpath3`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 13 and 14 only.

Integration 4 (intop4)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `Loaded edge integration` in the **Label** text field.

3 In the **Operator name** text field, type `LoadEdgeInt`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
5 Select Boundary 3 only.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.
2 In the **Settings** window for **Variables**, locate the **Variables** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `single_edge_crack_variables.txt`.

Use a fine mesh close to the crack tip where the stress gradients are large.

MESH 1

Free Triangular 1

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

Size 1

1 In the **Model Builder** window, right-click **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 **Domain**.
4 Select Domain 3 only.
5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

Size

1 In the **Model Builder** window, click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Predefined** list, choose **Fine**.
4 Click **Build All**.

Set up a parametric sweep over the crack length.

STUDY 1

Parametric Sweep

1 In 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
Xa (Crack length)	`range(0.5,da,0.7)`	`m`

5 In the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

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

2 From the **Parameter value (Xa (m))** list, choose **0.6**.

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

5 In the **Minimum** text field, type `0`.

6 In the **Maximum** text field, type `140`.

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

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

Global Evaluation 1

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Stress intensity factors` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

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

Expression	Unit	Description
`KI_1`		`Stress intensity factor, contour 1`
`KI_2`		`Stress intensity factor, contour 2`
`KI_3`		`Stress intensity factor, contour 3`

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

Compare J-integral by with an energy release rate based on numerical differentiation of the strain energy density with respect to the crack length.

RESULTS

1D Plot Group 3

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `J-integral and G` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter selection (Xa)** list, choose **Manual**.

5 In the **Parameter indices (1-21)** text field, type `range(2,20)`.

Global 1

1 Right-click **J-integral and G** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`J_1`	`J/m^2`	`J-integral, contour 1`
`((withsol('sol2',2*solid.Ws_tot,setval(Xa,Xa+da))-withsol('sol2',2*solid.Ws_tot,setval(Xa,Xa-da))))/(2*da*Th)`	`J/m^2`	`Energy release rate, strain energy`
`-((withsol('sol2',PE,setval(Xa,Xa+da))-withsol('sol2',PE,setval(Xa,Xa-da))))/(2*da*Th)`	`J/m^2`	`Energy release rate, load potential`

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

5 In the **Width** text field, type `2`.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

J-integral and G

1 In the **Model Builder** window, click **J-integral and G**.

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 `Crack length (m)`.

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

6 In the associated text field, type `Energy release rate (J/m^2)`.

7 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

8 In the **J-integral and G** toolbar, click **Plot**.

1D Plot Group 4

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

2 In the **Settings** window for **1D Plot Group**, type `Crack growth rate` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

Global 1

1 Right-click **Crack growth rate** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
`dadN`	`1`	`Crack growth rate (m/cycle)`

4 Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

Crack growth rate

1 In the **Model Builder** window, click **Crack growth rate**.

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 **x-axis label** check box.

5 In the associated text field, type `Crack length (m)`.

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

7 In the associated text field, type `Crack growth rate (m/cycle)`.

8 Locate the **Legend** section. Clear the **Show legends** check box.

9 In the **Crack growth rate** toolbar, click **Plot**.

Compute the total number of cycles needed for driving the crack from 0.5 m to 0.7 m.

Global Evaluation 2

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Number of cycles` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

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

Expression	Unit	Description
`1/dadN`	1	

5 Locate the **Data Series Operation** section. From the **Operation** list, choose **Integral**.

6 Click **Evaluate**.

Created in COMSOL Multiphysics 5.5

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 and friction forces. A boundary contact pair is created and the contact functionality in the Solid Mechanics interface is used to solve the contact problem. Both the penalty method and the augmented Lagrangian method are used, and friction is modeled with the Coulomb friction model.

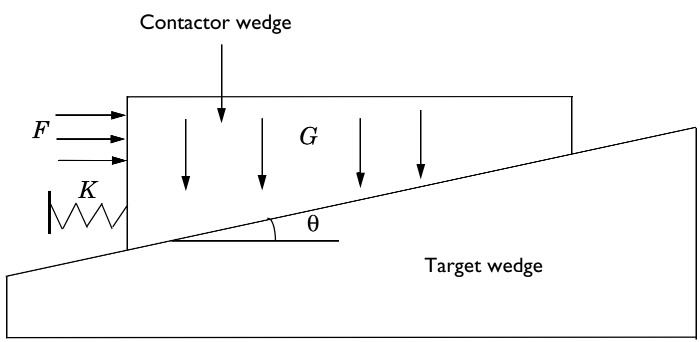


Figure 1: Sliding wedge with linear springs, a boundary load, and a gravity load.

The aim of this benchmark is to calculate the horizontal sliding distance and compare it with an elementary statics calculation. Three cases using different friction coefficients ($\mu = 0$; 0.1; 0.2) are analyzed.

For each friction coefficient, a specific total spring stiffness K is used (K = 1194 N/m; 882 N/m and 563.9 N/m respectively).

The horizontal applied force F = 1500 N, the total vertical gravity load G = 3058 N, the wedge angle is $\tan\theta = 0.1$.

For all study cases, the horizontal sliding distance is expected to be 1 m.

The mesh is shown in Figure 2.

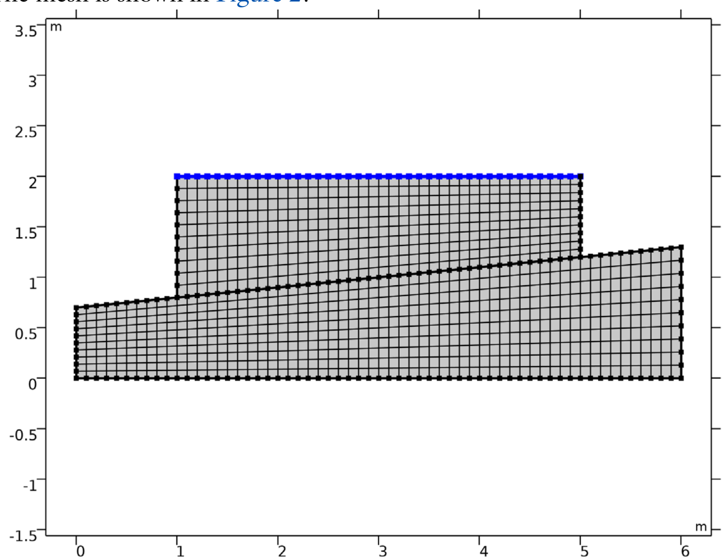


Figure 2: Quadrilateral elements are used to mesh the geometry.

The total number of elements in this model is 1000 and the number of degrees of freedom is 6484 for the displacement field.

Results and Discussion

The horizontal displacement computed for all friction cases agree very well with the reference data, see Ref. 1. For all cases, the difference is lower than 0.1%. Furthermore, both contact methods available in the Structural Mechanics Module converge to the same results. However, for this type of large sliding problem, the convergence and stability of the augmented Lagrangian method is superior to the penalty method.

Figure 3 below shows the result for the case $\mu = 0.2$, $K = 563.9$ N/m, and Figure 4 show the contact pressure and friction forces for the same case. Both figures shows the results obtained with the penalty method..

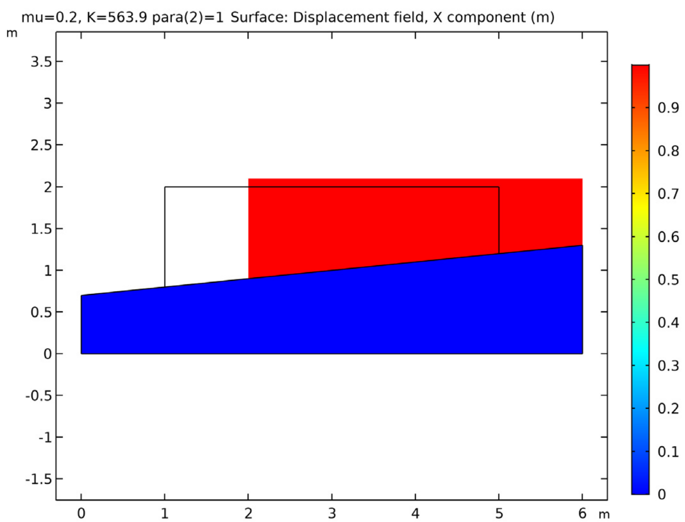


Figure 3: A surface plot of the x-displacement of the contactor wedge.

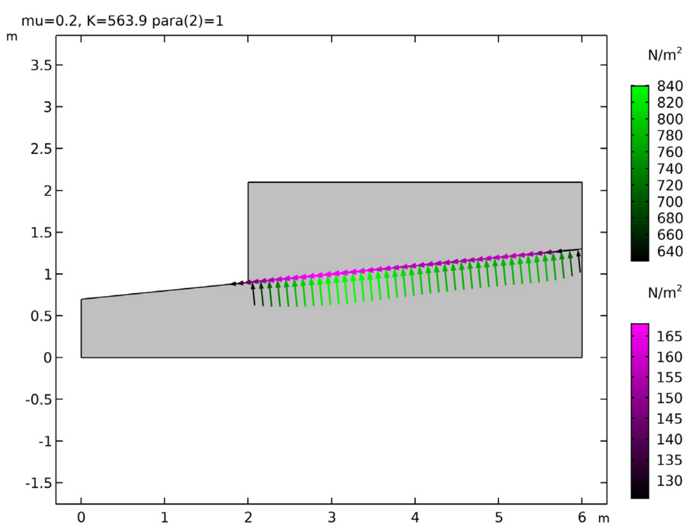


Figure 4: Contact pressure and friction forces acting on the contactor wedge.

Notes About the COMSOL Implementation

The initial unloaded state of the model is unstable and cause difficulties for the solver to find an initial solution. To avoid this issue, the first parameter step is set to 0.001. For this parameter value, a small amount of friction forces are present that stabilize the model.

The penalty method is not ideal for the type of large sliding problem with friction modeled in this example. While it in the limit will converge to the correct solution, the problem is stiff and ill-conditioned, meaning that small changes in the input can cause large changes to the results or even lead to no solution being found. In this example, the default solver suggestion does not give a stable solution, and the solver settings are modified to obtain a correct solution. Even with the modified settings, a warning from the linear solver gives an indication that the problem is ill-conditioned.

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 **General Studies>Stationary**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
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
para	0	0	Computation parameter

GEOMETRY 1

Polygon 1 (pol1)

1. In the **Geometry** toolbar, click **Polygon**.
2. In the **Settings** window for **Polygon**, locate the **Coordinates** section.
3. In the table, enter the following settings:

x (m)	y (m)
0	0
6	0
6	1.3
0	0.7

4. Click **Build All Objects**.

Rectangle 1 (r1)

1. In the **Geometry** toolbar, click **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 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.
2 Select the object **pol1** only.
3 In the **Settings** window for **Copy**, click **Build Selected**.

Difference 1 (dif1)

1 In 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 **Activate selection** toggle button.
5 Select the object **copy1** only.
6 Click **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 Click **Build Selected**.
6 Click the **Zoom Extents** button in 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	Variable	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/m^3	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 **Physics** toolbar, in the **Boundary** section, click **Pairs** and choose **Contact**.

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

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (ap1)** in the **Pairs** list.

5 Click **OK**.

Friction 1

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

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

3 In the μ text field, type `mu`.

4 Locate the **Initial Value** section. From the **Previous contact state** list, choose **In contact**.

Spring Foundation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **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 **Total spring constant**.

5 From the list, choose **Diagonal**.

6 In the $\mathbf{k}_{tot}$ table, enter the following settings:

K	0
0	0

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** 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 $\mathbf{F}_{\mathrm{tot}}$ vector as

F*para	x
0	y

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 2 only.

MESH 1

Mapped 1

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

Distribution 1

1 In the **Model Builder** window, right-click **Mapped 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 10.

Distribution 2

1 In the **Model Builder** window, 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 In 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 (Friction coefficient)	`0 0.1 0.2`	

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
K (Spring stiffness)	`1194 882 563.9`	`N/m`

Step 1: Stationary

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

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 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 (Computation parameter)	`1e-3 1`	

Set a stricter tolerance and tune the parameter stepping of the auxiliary sweep to improve the convergence of the model. The convergence is also improved by changing the non-linear solver to Constant Newton.

Solution 1 (sol1)

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

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

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.
4 In the **Relative tolerance** text field, type `1e-6`.
5 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.
6 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.
7 Select the **Tuning of step size** check box.
8 In the **Initial step size** text field, type `1e-2`.
9 In the **Minimum step size** text field, type `1e-6`.
10 From the **Predictor** list, choose **Linear**.
11 In the **Model Builder** window, click **Fully Coupled 1**.
12 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
13 From the **Nonlinear method** list, choose **Constant (Newton)**.
14 In the **Model Builder** window, click **Study 1**.
15 In the **Settings** window for **Study**, type `Study 1: Penalty` in the **Label** text field.
16 In the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

In the **Settings** window for **2D Plot Group**, type `Displacement (solid): Penalty` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Displacement (solid): Penalty** 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 - m>u - Displacement field, X component**.
3 In the **Displacement (solid): Penalty** toolbar, click **Plot**.
4 Click the **Zoom Extents** button in the **Graphics** toolbar.

Applied Loads (solid)

1 In the **Model Builder** window, click **Applied Loads (solid)**.
2 In the **Settings** window for **Group**, type `Applied Loads (solid): Penalty` in the **Label** text field.

Contact (solid)

1 In the **Model Builder** window, click **Contact (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Contact (solid): Penalty` in the **Label** text field.

Gray Surfaces

1 In the **Model Builder** window, expand the **Results>Contact (solid): Penalty** node.

2 Right-click **Gray Surfaces** and choose **Enable**.

3 In the **Contact (solid): Penalty** toolbar, click **Plot**.

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

Follow the instructions below to evaluate the horizontal displacement for all three friction cases.

Point Evaluation 1

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

2 In the **Settings** window for **Point Evaluation**, type `Point Evaluation 1: Penalty` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1: Penalty/ 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 - m>u - Displacement field, X component**.

8 Click **Evaluate**.

Now, solve the model using the augmented Lagrangian contact method.

SOLID MECHANICS (SOLID)

Contact 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Contact 1** and choose **Duplicate**.

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

3 From the **Formulation** list, choose **Augmented Lagrangian**.

ADD STUDY

1 In 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 **General Studies>Stationary**.

4 Click **Add Study**.

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

STUDY 2

Parametric Sweep

1 In 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 (Friction coefficient)	`0 0.1 0.2`	

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
K (Spring stiffness)	`1194 882 563.9`	`N/m`

Step 1: Stationary

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

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

3 Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Solid Mechanics (solid), Controls spatial frame>Contact 1**.

5 Click **Disable**.

6 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Computation parameter)	`1e-3 1`	

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

10 In the **Settings** window for **Study**, type `Study 1: Augmented Lagrangian` in the **Label** text field.

In this example the contact forces are very small, so it is necessary so set proper scales for these variables.

Solution 6 (sol6)

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

2 In the **Model Builder** window, expand the **Solution 6 (sol6)** node.

3 In the **Model Builder** window, expand the **Study 1: Augmented Lagrangian>Solver Configurations>Solution 6 (sol6)>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, 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: Augmented Lagrangian>Solver Configurations>Solution 6 (sol6)>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 `0.1`.

13 In the **Maximum step size** text field, type `1`.

14 In the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

In the **Settings** window for **2D Plot Group**, type `Displacement (solid): Augmented Lagrange` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Displacement (solid): Augmented Lagrange** 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 - m>u - Displacement field, X component**.

3 In the **Displacement (solid): Augmented Lagrange** toolbar, click **Plot**.

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

Applied Loads (solid)

1 In the **Model Builder** window, click **Applied Loads (solid)**.

2 In the **Settings** window for **Group**, type `Applied Loads (solid): Augmented Lagrange` in the **Label** text field.

Contact (solid)

1 In the **Model Builder** window, click **Contact (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Contact (solid): Augmented Lagrange` in the **Label** text field.

Gray Surfaces

1 In the **Model Builder** window, expand the **Results>Contact (solid): Augmented Lagrange** node.

2 Right-click **Gray Surfaces** and choose **Enable**.

3 In the **Contact (solid): Augmented Lagrange** toolbar, click **Plot**.

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

Point Evaluation 1: Penalty 1

1 In the **Model Builder** window, right-click **Point Evaluation 1: Penalty** and choose **Duplicate**.

2 In the **Settings** window for **Point Evaluation**, type `Point Evaluation 1: Augmented Lagranage` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1: Augmented Lagrangian/Parametric Solutions 2 (sol7)**.

4 Click **New Table**.

Prepare the model for later use by disabling the second contact feature in the first study (Penalty).

STUDY 1: PENALTY

Step 1: Stationary

1. In the **Model Builder** window, under **Study 1: Penalty** click **Step 1: Stationary**.
2. In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3. Select the **Modify model configuration for study step** check box.
4. In the **Physics and variables selection** tree, select **Component 1 (comp1)> Solid Mechanics (solid), Controls spatial frame>Contact 2**.
5. Click **Disable**.

Created in COMSOL Multiphysics 5.5

Instability of a Space Arc Frame

Model Definition

In this example you study the lateral deflection of a space frame subjected to concentrated vertical loading at four different points. A small lateral load is applied to break the symmetry of the structure. The model is described in detail in section 6.3 of Ref. 1, where it is called “Space frame subjected to concentrated loading”. A schematic description of the frame and loads are shown in Figure 1. There are two types of members used in the frame, marked as 1 and 2 respectively.

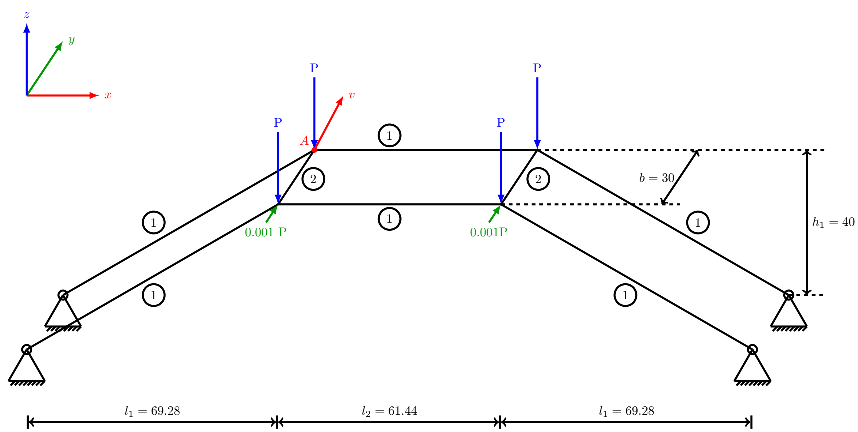


Figure 1: Space frame geometry.

GEOMETRY

- Cross section properties of type 1 members are A_1 = 0.5, I_{y1} = 0.4, I_{z1}=0.133.
- Cross section properties of type 2 members are A_2 = 0.1, I_{y2} = 0.05, I_{z2}=0.05.

The local y direction coincides with the global y direction.

The torsional constant is not supplied in the reference, so the common approximation J = $I_y + I_z$ is used.

MATERIAL

Linear elastic with $E = 4.32 \cdot 10^5$ and G = $1.66 \cdot 10^5$.

CONSTRAINTS AND LOADS

- All the base points of the frame are pinned.
- The four corners at the top are subjected to vertical loads P, ranging from 0 to 8.65, acting downward.

- The front two corners are subjected to lateral loads of $0.001 \cdot P$.

Results and Discussion

With only vertical loads active on the frame this is a symmetric problem. Hence, it is necessary to perturb the symmetry somewhat to induce a controlled instability. The small lateral loads serve this purpose. As an alternative, you could introduce an initial imperfection in the geometry.

Figure 2 below shows the final state of the frame.

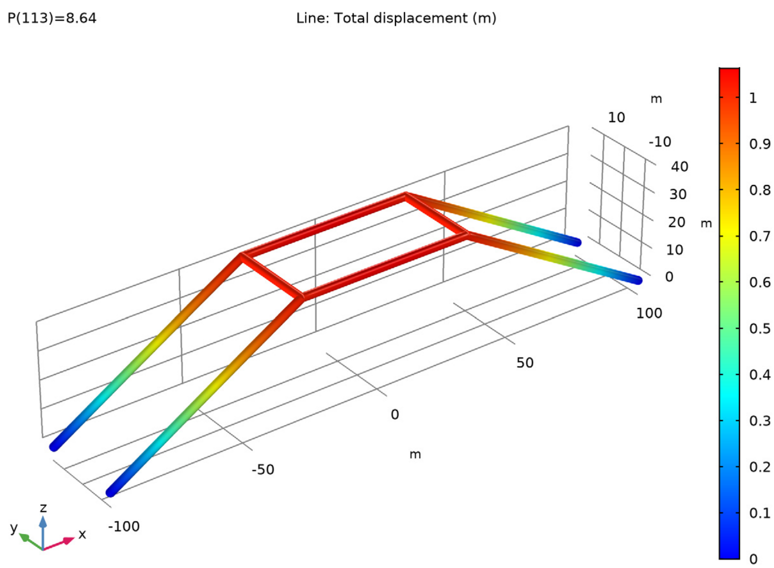


Figure 2: Final state of the deformed frame.

The horizontal displacement of point A on the frame versus the compressive load is shown in Figure 3. Data obtained from Ref. 1 is marked on the same curve. The agreement with the data from the reference is very good.

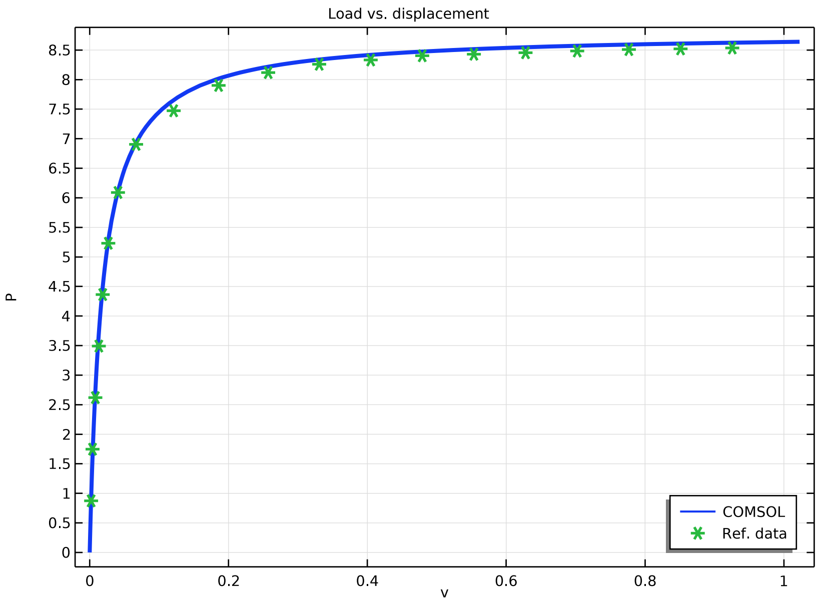


Figure 3: Load vs. displacement.

The plot of the lateral deflection shows that an instability occurs at a parameter value close to 8.0. In practice, the critical load of an imperfect structure is often significantly lower than that of the ideal structure.

Linear buckling analysis also gives the first critical buckling load as 8.67 which matches well with the critical load obtained from the above analysis. Corresponding buckling mode shape is shown in the Figure 4 below.

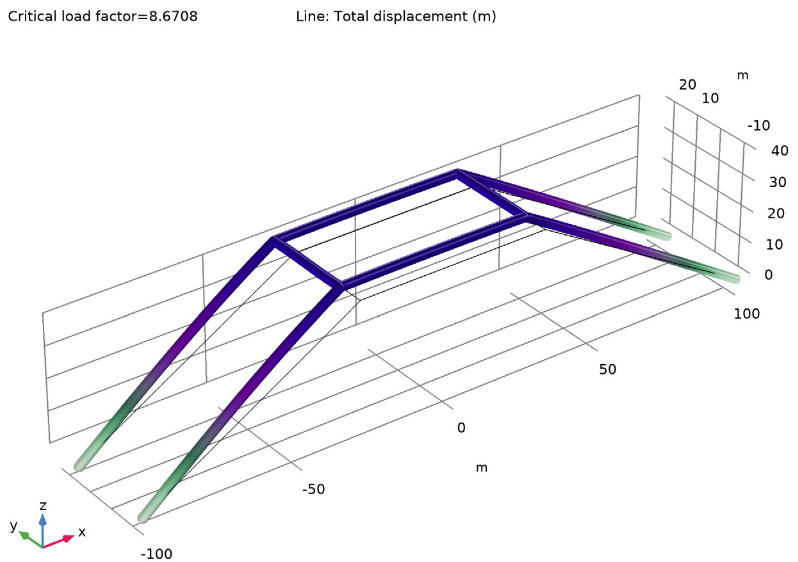


Figure 4: First buckling mode.

Reference

1. Z.X. Li and L. Vu-Quoc, *A Mixed Co-rotational 3D Beam Element for Arbitrarily Large Rotations*, Advanced Steel Construction Vol. 6, No. 2, 767-787, 2010.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/space_frame_instability`

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 **General Studies>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Define the load parameter as well as the geometric data.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `space_frame_instability_parameters.txt`.

GEOMETRY 1

Since the frame is symmetric, create only one quarter of the geometry and use two mirror operations to obtain the full geometry.

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.
2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.
3 From the **Data source** list, choose **Vectors**.
4 In the **x** text field, type `-l1-l2/2 -l2/2 -l2/2 0 `.
5 In the **y** text field, type `-b/2 -b/2 -b/2 -b/2`.
6 In the **z** text field, type `0 h1 h1 h1`.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.
2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.
3 From the **Specify** list, choose **Coordinates**.
4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
5 Locate the **Starting Point** section. In the **x** text field, type `-l2/2`, **y** to `-b/2`, and **z** to `h1`.

6 Locate the **Endpoint** section. In the **x** text field, type `-l2/2` and **z** to `h1`.

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

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

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Point on Plane of Reflection** section. In the **x** text field, type `-l2/2`.

6 In the **z** text field, type `h1`.

7 Locate the **Normal Vector to Plane of Reflection** section. In the **y** text field, type `1`.

8 In the **z** text field, type `0`.

Mirror 2 (mir2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

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

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Normal Vector to Plane of Reflection** section. In the **x** text field, type `1`.

6 In the **z** text field, type `0`.

7 In the **Geometry** toolbar, click **Build All**.

8 Click the **Go to Default View** button in the **Graphics** toolbar.

BEAM (BEAM)

Linear Elastic Material 1

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

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

3 From the **Specify** list, choose **Young's modulus and shear modulus**.

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	Variable	Value	Unit	Property group
Young's modulus	E	4.32e5	Pa	Basic
Shear modulus	G	1.66e5	N/m²	Bulk modulus and shear modulus
Density	rho	0	kg/m³	Basic

The density is set to zero since it is not used in the present analysis.

BEAM (BEAM)

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 **Basic Section Properties** section.

3 In the A text field, type `A1`.

4 In the I_{zz} text field, type `Iz1`.

5 In the I_{yy} text field, type `Iy1`.

6 In the J text field, type `Iy1+Iz1`.

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

Cross Section Data 2

1 In the **Physics** toolbar, click **Edges** and choose **Cross Section Data**.

2 Select Edges 3, 5, 9, and 11 only.

3 In the **Settings** window for **Cross Section Data**, locate the **Basic Section Properties** section.

4 In the A text field, type `A2`.

5 In the I_{zz} text field, type `Iz2`.

6 In the I_{yy} text field, type `Iy2`.

7 In the J text field, type `Iy2+Iz2`.

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 From the **Orientation method** list, choose **Orientation vector**.

4 Specify the V vector as

1	X
0	Y
0	Z

Pinned 1

1 In the **Physics** toolbar, click **Points** and choose **Pinned**.

2 Select Points 1, 2, 11, and 12 only.

Point Load 1

1 In the **Physics** toolbar, click **Points** and choose **Point Load**.

2 Select Points 3, 5, 8, and 10 only.

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	z

Point Load 2

1 In the **Physics** toolbar, click **Points** and choose **Point Load**.

2 Select Points 3 and 8 only.

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

4 Specify the $\mathbf{F}_P$ vector as

0	x
0.001*P	y
0	z

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

STUDY 1

Step 1: Stationary

Use geometric nonlinearity since the problem is expected to have an instability.

1 In the **Model Builder** window, under **Study 1** 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 parametric sweep for the load.

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

5 Click **Add**.

Due to instability, the load increment for **P**>8 is reduced.

6 In the table, enter the following settings:

Parameter name	Parameter value list
P (Load)	range(0,0.1,8) range(8.02, 0.02, 8.65)

Solution 1 (sol1)

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

Scale the dependent variables appropriately.

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 (comp1.beam.uLin)**.

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

5 From the **Method** list, choose **Manual**.

6 In the **Model Builder** window, click **Rotation field (comp1.beam.thLin)**.

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 `pi/10`.

Increase the maximum allowed number of iterations due to the expected instability.

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

11 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

12 In the **Maximum number of iterations** text field, type `40`.

13 In the **Study** toolbar, click **Compute**.

RESULTS

Stress (beam)

In the **Settings** window for **3D Plot Group**, type `Displacement (beam)` in the **Label** text field.

Line 1

1 In the **Model Builder** window, expand the **Results>Displacement (beam)** 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 - m**.

3 In the **Displacement (beam)** toolbar, click **Plot**.

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

Compare load-displacement curve with values from the reference.

1D Plot Group 10

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

2 In the **Settings** window for **1D Plot Group**, type `Load vs. displacement` in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **x-axis label** check box.

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

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

6 In the associated text field, type `P`.

7 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

8 In the **Title** text area, type `Load vs. displacement`.

Point Graph 1

1 Right-click **Load vs. displacement** and choose **Point Graph**.

2 Select Point 4 only.

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

4 In the **Expression** text field, type `P`.

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

6 In the **Expression** text field, type `beam.uLinY`.

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
COMSOL

10 Click to expand the **Coloring and Style** section. In the **Width** text field, type `3`.

Table 1

1 In the **Results** toolbar, click **Table**.

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

3 Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `space_frame_instability_data.txt`.

5 In the **Label** text field, type `Ref data`.

Table Graph 1

1 In the **Model Builder** window, right-click **Load vs. displacement** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 In the **Number** text field, type `20`.

6 Click to expand 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
`Ref. data`

Load vs. displacement

1 In the **Model Builder** window, click **Load vs. displacement**.

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

3 From the **Position** list, choose **Lower right**.

4 In the **Load vs. displacement** toolbar, click **Plot**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Next, you verify the critical buckling load by performing the **linear buckling** analysis.

ROOT

In the **Home** toolbar, click **Windows** and choose **Add Study**.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Linear Buckling**.

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

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

STUDY 2

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

First default plot from the buckling analysis shows the first buckling mode shape as shown in Figure 4.

RESULTS

Mode Shape (beam)

1 In the **Mode Shape (beam)** toolbar, click **Plot**.

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

Spherical Cap with Central Point Load

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 spherical cap with a point load at its crown is a common example to study postbuckling analysis of 2D axisymmetric shells. The critical load, snap-through behavior, softening and stiffing effects are the interesting aspects which are studied in this example.

In order to predict the postbuckling behavior, one need to 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 phenomenon cannot be always simulated using prescribed load in a standard nonlinear static solver because the problem becomes numerically singular. In the current example, the displacement at the crown increases monotonically even if the load decreases after a critical point in the snap-through region. Thus, using displacement control is a useful strategy for this example.

Model Definition

The model studied here is a benchmark for a spherical cap subjected to a point load at its crown; see Ref. 1.

- The radius of the spherical cap is $a = 10$ m and the thickness is $th = 0.20384$ m. The sector angle of the spherical cap is $\pi/4$ radians.
- The edge/point which is not on axis of revolution is fixed.
- In the study the variation of the crown (center) axial displacement with respect to the applied load is of interest.

Due to the axial symmetry, only the part of the cap which is located at positive r-coordinates is modeled. The full geometry of the spherical cap with loading and boundary conditions is shown in Figure 1.

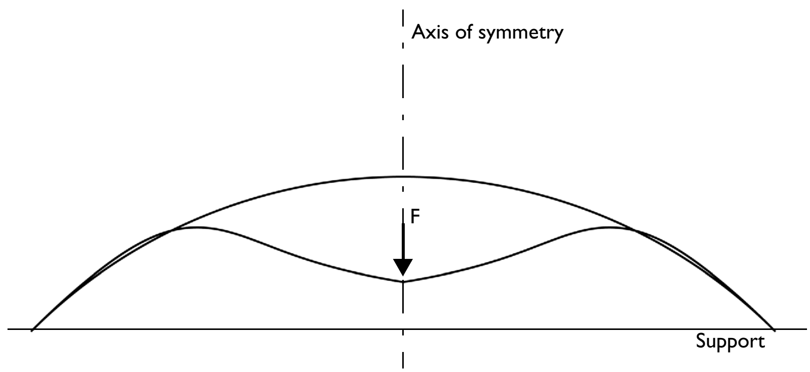


Figure 1: Problem description.

Results

For a spherical cap, the load versus displacement curve exhibits a critical load which is followed by a gradual snap and further increase in stiffness. Figure 2 and Figure 3 show the total displacement using the Solid Mechanics and Shell interfaces, respectively, at three different crown displacements. The annotations in the figures shows the corresponding point loads which closely match the benchmarked numerical solutions given in Ref. 1.

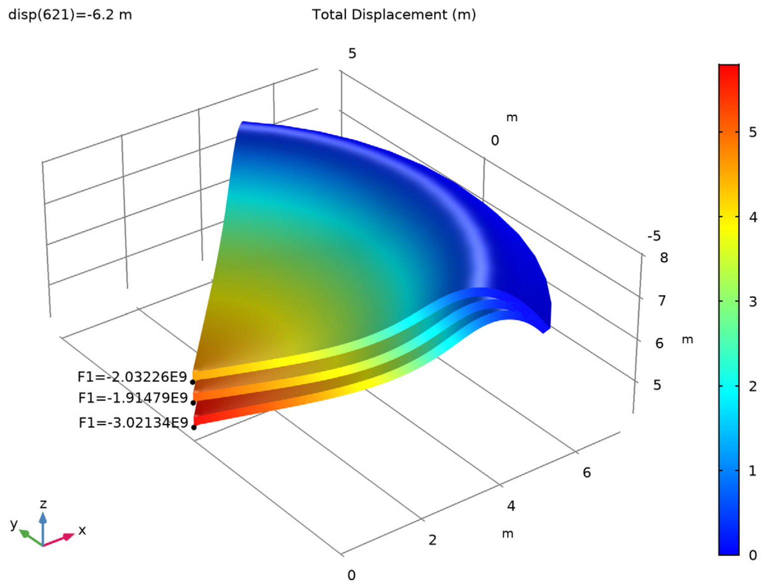


Figure 2: Total displacement computed in the Solid Mechanics Interface using 40 mesh elements.

What's important to note in the figures is the snap-through behavior and softening effect after the critical load. The top surface in both figures corresponds to the critical load, while the middle surface is corresponding to the load after the critical point. This shows that although deformation increases the load decreases due to softening after the critical load. The third surface in both figures shows an increase in displacement with an increase in load, indicating an increase in stiffness after the snap through phase.

Figure 4 shows the variation of axial displacement at the crown of the spherical cap versus the applied load. For the Shell interface, three different discretizations (4, 8, 16 mesh elements) are used. For the Solid Mechanics interface 40 mesh elements are used. These discretizations are the same as in Ref. 1.

The results match the values in the reference quite closely. Note however, that these results are reported for certain discretizations and element formulations. There is no target value as such.

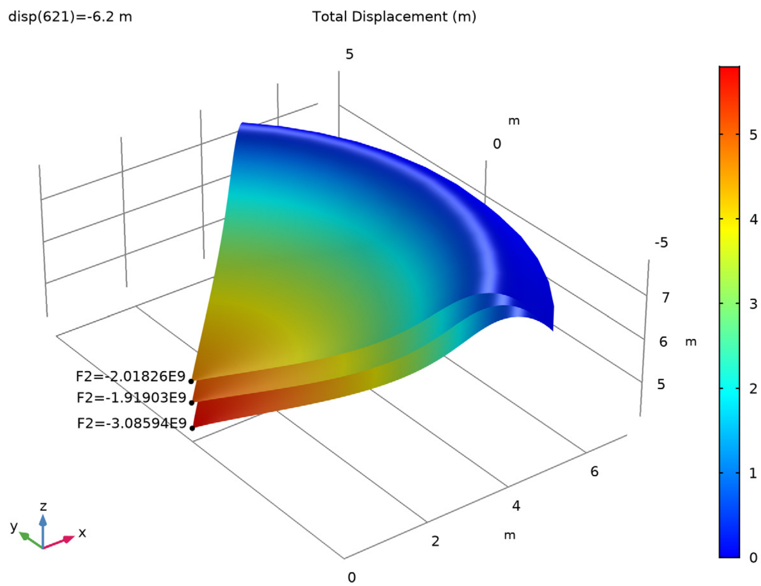


Figure 3: Total displacement computing in the Shell Interface using 16 mesh elements.

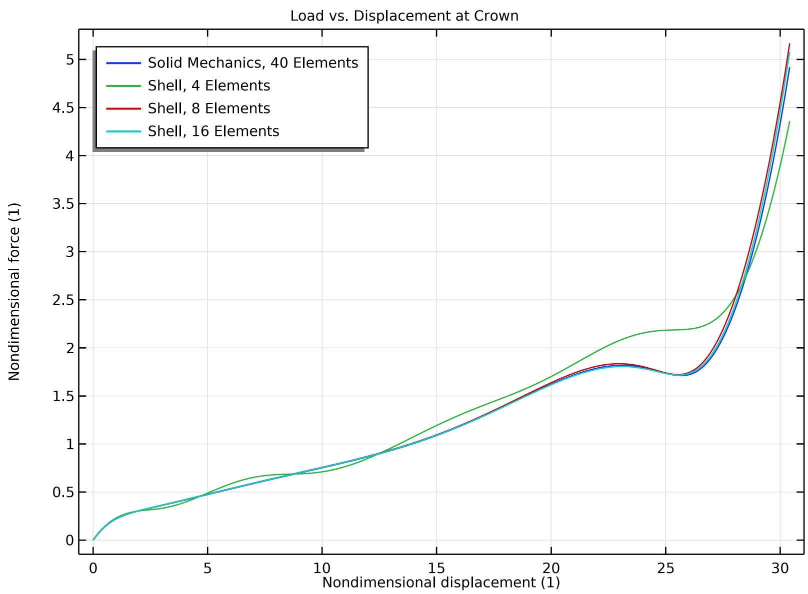


Figure 4: Applied load versus center displacement.

In Table 1, the results from the Solid Mechanics interface with 40 mesh elements are compared with the reference.

TABLE 1: SOLID MECHANICS IN NONDIMENSIONAL FORMAT.

Applied Load	**Displacement in reference**	**Displacement computed**
0.320	2.165	2.250
0.584	6.769	6.920
0.975	13.335	13.600
1.624	19.706	20.025
1.808	22.073	22.450
1.758	24.398	24.665
1.962	26.788	27.170
4.699	29.851	30.265

In Table 2, Table 3, and Table 4, the results from the Shell interface with 4, 8 and 16 mesh elements respectively, are compared with the reference. Note that with only four elements, there is no snap through behavior, indicating that the mesh is much to coarse. This is experienced also in the reference, even though different types of shell element formulations are used.

TABLE 2: SHELL RESULTS WITH 4 ELEMENTS IN NONDIMENSIONAL FORMAT.

Applied Load	Displacement target	Displacement computed
0.335	2.367	3.100
0.579	6.921	5.940
0.920	11.614	12.665
1.176	16.423	14.850
1.705	18.964	20.300
2.488	21.393	27.850
2.540	23.659	28.050
3.765	28.541	29.870

TABLE 3: SHELL RESULTS WITH 8 ELEMENTS IN NONDIMENSIONAL FORMAT.

Applied Load	Displacement target	Displacement computed
0.332	2.326	2.440
0.580	6.720	6.775
0.994	13.642	13.760
1.502	18.487	18.815
1.757	20.887	21.240
1.678(1.722)	25.668	25.500
3.705	28.680	29.330

TABLE 4: SHELL RESULTS WITH 16 ELEMENTS IN NONDIMENSIONAL FORMAT.

Applied Load	Displacement target	Displacement computed
0.332	2.326	2.445
0.580	6.720	6.800
0.994	13.642	13.800
1.502	18.487	18.945
1.757	20.887	21.640
1.678(1.717)	25.668	25.500
3.705	28.680	29.410

Note that the lowest load after the critical load when using a shell formulation is 1.678 in the reference. This value is not reached in the solutions, where the lowest load is predicted as 1.722 and 1.717 with 8 and 16 elements, respectively. A refined Solid Mechanics model actually indicates that the current that the values computed here are more accurate than those reported in the reference.

Notes About the COMSOL Implementation

The main feature of this model is that a limit point instability occurs at the buckling load. Load control would not able to track the unstable solution paths after the limit point, so a displacement control is used since the displacement at the crown increases monotonically.

In this case, where the only load is a point load, it would be possible to directly prescribe the displacement in that point, and then measure the reaction force. If the load was more complex, for example a pressure load, that would not be possible. For this reason, a more general approach is shown here.

To employ a displacement control strategy, a point load at the crown is considered as a global degree of freedom and a global equation in terms of axial displacement at the crown is solved to get the point load value.

For a nonlinear problem experiencing a snap-through behavior there is no general way to determine which controlling parameter to use, so it is necessary to use some physical insight. You need to find a quantity which is monotonically increasing to use as a controlling parameter.

Reference

1. P. Lyons and S. Holsgrove, *Finite Element Benchmarks For 2D Beams And Axisymmetric Shells Involving Geometric Non-Linearity,* NAFEMS, 2005.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/spherical_cap_with_central_point_load`

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 In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.

5 Click **Add**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
a	10[m]	10 m	Radius of cap
th	0.203840[m]	0.20384 m	Thickness of cap
EE	210e9[Pa]	2.1E11 Pa	Young's modulus
Nu	0.3	0.3	Poisson's ratio
Rho	7800	7800	Density
disp	0[m]	0 m	Displacement parameter
meshdist	4	4	Mesh distribution parameter

Define a set of nondimensional variables that will be useful in the postprocessing plots and evaluations.

DEFINITIONS

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`Fn1`	`-F1*a/(EE*th^3*2*pi)`		`Nondimensional force`
`wn1`	`-w/th`		`Nondimensional displacement`
`Fn2`	`-F2*a/(EE*th^3*2*pi)`		`Nondimensional force`
`wn2`	`-w2/th`		`Nondimensional displacement`

GEOMETRY 1

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Sector angle** text field, type `45`.

5 In the **Radius** text field, type `a+th`.

6 Click **Build Selected**.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `45`.

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

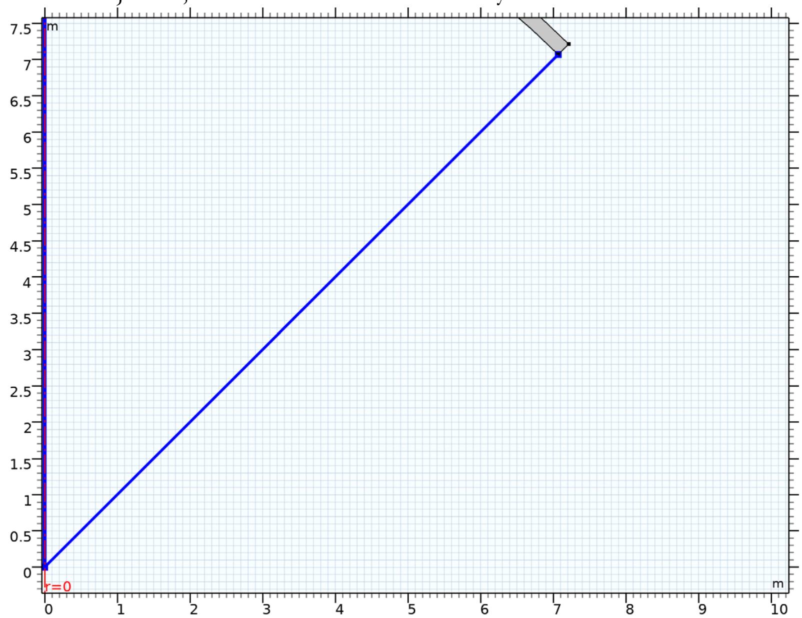
Layer name	Thickness (m)
`Layer 1`	`th`

9 Click **Build Selected**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **c1**, select Boundaries 1 and 2 only.



3 In the **Settings** window for **Delete Entities**, click **Build Selected**.

Add a same material through a material link for **Solid Mechanics** and **Shell** interfaces.

GLOBAL DEFINITIONS

Material 1 (mat1)

In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

MATERIALS

Material Link 1 (matlnk1)

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

Material Link 2 (matlnk2)

1 Right-click **Materials** and choose **More>Material Link**.

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

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

4 Select Boundary 3 only.

It might be easier to select the correct boundary by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)

GLOBAL DEFINITIONS

Material 1 (mat1)

1 In the **Model Builder** window, under **Global Definitions>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	Variable	Value	Unit	Property group
Young's modulus	E	EE	Pa	Basic
Poisson's ratio	nu	Nu	1	Basic
Density	rho	Rho	kg/m³	Basic

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal 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 1 only.

5 Locate the **Advanced** section. From the **Method** list, choose **Summation over nodes**.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 Select Boundary 2 only.

Now add a global equation for a point load, so that the crown displacement equals to the prescribed one. For that, you need to show advanced physics options.

3 Click the **Show More Options** button in the **Model Builder** toolbar.

4 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

5 Click **OK**.

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **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
`F1`	`intop1(w)-disp`	`0`	`0`	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `force` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Force (N)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `disp` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Displacement (m)**.

14 Click **OK**.

Point Load (on Axis) 1

1 In the **Physics** toolbar, click **Points** and choose **Point Load (on Axis)**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Load (on Axis)**, locate the **Force** section.

4 From the F_z list, choose **State variable F1 (solid/ge1)**.

SHELL (SHELL)

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

2 In the **Settings** window for **Shell**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 3 only.

In order to model the solid midplane using Shell interface assign a proper offset from the **Thickness and Offset** feature. As shell normal is pointing inward (which can be verified in postprocessing plot) use `-th/2` as a physical offset.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Thickness and Offset 1**.
2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.
3 In the d text field, type `th`.
4 From the **Offset definition** list, choose **Physical offset**.
5 In the z_{offset} text field, type `-th/2`.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.
2 Select Point 3 only.

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **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**
F2	intop1(w2)-disp	0	0	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.
5 In the **Physical Quantity** dialog box, type `force` in the text field.
6 Click **Filter**.
7 In the tree, select **General>Force (N)**.
8 Click **OK**.
9 In the **Settings** window for **Global Equations**, locate the **Units** section.
10 Click **Select Source Term Quantity**.
11 In the **Physical Quantity** dialog box, type `disp` in the text field.
12 Click **Filter**.
13 In the tree, select **General>Displacement (m)**.
14 Click **OK**.

Point Load (on Axis) 1

1 In the **Physics** toolbar, click **Points** and choose **Point Load (on Axis)**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Load (on Axis)**, locate the **Force** section.

4 From the F_z list, choose **State variable F2 (shell/ge1)**.

Use different **Mesh** nodes in order to use different discretizations for Solid Mechanics and Shell interfaces as given in the benchmark example.

MESH 2

In the **Mesh** toolbar, click **Add Mesh**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, type `Mesh: Solid Mechanics` in the **Label** text field.

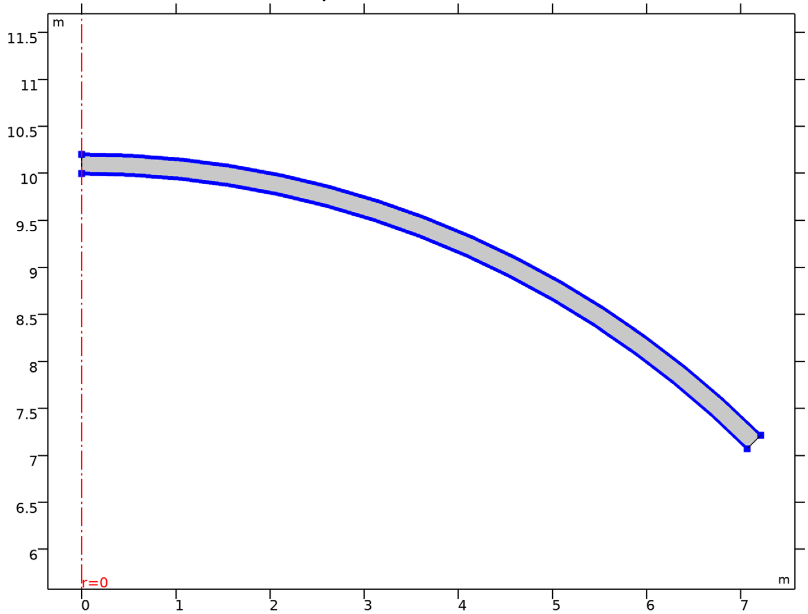
Mapped 1

Right-click **Component 1 (comp1)>Meshes>Mesh: Solid Mechanics** and choose **Mapped**.

Distribution 1

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 3 and 4 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 Selected**.

MESH 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes** click **Mesh 2**.

2 In the **Settings** window for **Mesh**, type `Mesh: Shell` in the **Label** text field.

Edge 1

1 Right-click **Component 1 (comp1)>Meshes>Mesh: Shell** and choose **More Operations> Edge**.

2 Select Boundary 3 only.

Distribution 1

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

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

3 In the **Number of elements** text field, type `meshdist`.

4 Click **Build Selected**.

Add a stationary study for **Solid Mechanics** interface.

ROOT

In the **Home** toolbar, click **Windows** and choose **Add Study**.

ADD STUDY

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

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

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Shell (shell)** interface.

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

STUDY 1

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

2 In the **Settings** window for **Study**, type `Study: Solid Mechanics` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Solid Mechanics** 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 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Geometry	Mesh
Geometry 1	Mesh: Solid Mechanics

5 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
disp (Displacement parameter)	`range(0,-0.01,-6.2)`	`m`

8 In the **Home** toolbar, click **Compute**.

Add a stationary study for **Shell** interface. Parameterize the mesh discretization using a parametric sweep.

ROOT

Click **Windows** and choose **Add Study**.

ADD STUDY

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

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

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Solid Mechanics (solid)** interface.

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

STUDY 2

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

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

Parametric Sweep

1 In 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
meshdist (Mesh distribution parameter)	`4,8,16`	

Step 1: Stationary

1 In the **Model Builder** window, 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 **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
disp (Displacement parameter)	`range(0,-0.01,-6.2)`	`m`

7 In the **Study** toolbar, click **Compute**.

RESULTS

Revolution 2D 1

1 In the **Model Builder** window, expand the **Datasets** node, then click **Revolution 2D 1**.

2 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution Layers** section.

3 In the **Start angle** text field, type 45.

4 In the **Revolution angle** text field, type -90.

Revolution 2D 2

1 In the **Model Builder** window, click **Revolution 2D 2**.

2 In the **Settings** window for **Revolution 2D**, locate the **Revolution Layers** section.

3 In the **Start angle** text field, type 45.

4 In the **Revolution angle** text field, type -90.

In order to visualize the softening and stiffening effect after the critical point, generate a 3D displacement plot of spherical cap at the critical point, and on unstable and stable part of the equilibrium path after critical point.

Stress, 3D (solid)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Total Displacement, 3D (solid)` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Total Displacement (m)`.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1 In the **Model Builder** window, expand the **Results>Total Displacement, 3D (solid)** node, then click **Surface 1**.

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

3 From the **Dataset** list, choose **Revolution 2D 1**.

4 From the **Parameter value (disp (m))** list, choose **-4.7**.

5 Locate the **Expression** section. In the **Expression** text field, type `solid.disp`.

Annotation 1

1 In the **Model Builder** window, expand the **Surface 1** node.

2 Right-click **Total Displacement, 3D (solid)** and choose **Annotation**.

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

4 From the **Dataset** list, choose **Revolution 2D 1**.

5 From the **Parameter value (disp (m))** list, choose **-4.7**.

6 Locate the **Annotation** section. In the **Text** text field, type `F1=eval(F1)`.

7 Select the **Allow evaluation of expressions** check box.

8 From the **Geometry level** list, choose **Global**.

9 Locate the **Position** section. In the **Z** text field, type `a-4.7`.

10 Click to expand the **Advanced** section. Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Lower right**.

Surface 2

In the **Model Builder** window, under **Results>Total Displacement, 3D (solid)** right-click **Surface 1** and choose **Duplicate**.

Annotation 2

In the **Model Builder** window, under **Results>Total Displacement, 3D (solid)** right-click **Annotation 1** and choose **Duplicate**.

Surface 2

1 In the **Settings** window for **Surface**, locate the **Data** section.

2 From the **Parameter value (disp (m))** list, choose **-5.2**.

3 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Annotation 2

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

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

3 From the **Parameter value (disp (m))** list, choose **-5.2**.

4 Locate the **Position** section. In the **Z** text field, type `a-5.2`.

Surface 3

In the **Model Builder** window, under **Results>Total Displacement, 3D (solid)** right-click **Surface 2** and choose **Duplicate**.

Annotation 3

In the **Model Builder** window, under **Results>Total Displacement, 3D (solid)** right-click **Annotation 2** and choose **Duplicate**.

Surface 3

1 In the **Settings** window for **Surface**, locate the **Data** section.

2 From the **Parameter value (disp (m))** list, choose **-5.8**.

Annotation 3

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

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

3 From the **Parameter value (disp (m))** list, choose **-5.8**.

4 Locate the **Position** section. In the **Z** text field, type `a-5.8`.

5 In the **Total Displacement, 3D (solid)** toolbar, click **Plot**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

For better visualization change the dataset from **Shell 1** to **Study: Shell/Solution 2**.

Stress (shell)

1 In the **Model Builder** window, click **Stress (shell)**.

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

3 From the **Dataset** list, choose **Study: Shell/Solution 2 (sol2)**.

Line 1

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

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

3 In the **Expression** text field, type `shell.mises_max`.

4 Locate the **Coloring and Style** section. In the **Radius scale factor** text field, type `1`.

For better visualization change the dataset in **Revolution 2D 2** from **Shell 1** to **Study: Shell/ Solution 2**.

Revolution 2D 2

1 In the **Model Builder** window, click **Revolution 2D 2**.

2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study: Shell/Solution 2 (sol2)**.

Stress, 3D (shell)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (shell)**.

2 In the **Settings** window for **3D Plot Group**, type `Total Displacement, 3D (shell)` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Total Displacement (m)`.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1 In the **Model Builder** window, expand the **Results>Total Displacement, 3D (shell)** node, then click **Surface 1**.

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

3 From the **Dataset** list, choose **Revolution 2D 2**.

4 From the **Parameter value (disp (m))** list, choose **-4.7**.

5 Locate the **Expression** section. In the **Expression** text field, type `shell.disp`.

Annotation 1

1 In the **Model Builder** window, expand the **Surface 1** node.

2 Right-click **Total Displacement, 3D (shell)** and choose **Annotation**.

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

4 From the **Dataset** list, choose **Revolution 2D 2**.

5 From the **Parameter value (disp (m))** list, choose **-4.7**.

6 Locate the **Annotation** section. In the **Text** text field, type `F2=eval(F2)`.

7 Select the **Allow evaluation of expressions** check box.

8 From the **Geometry level** list, choose **Global**.

9 Locate the **Position** section. In the **Z** text field, type `a-4.7`.

10 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Lower right**.

Surface 2

In the **Model Builder** window, under **Results>Total Displacement, 3D (shell)** right-click **Surface 1** and choose **Duplicate**.

Annotation 2

In the **Model Builder** window, under **Results>Total Displacement, 3D (shell)** right-click **Annotation 1** and choose **Duplicate**.

Surface 2

1 In the **Settings** window for **Surface**, locate the **Data** section.

2 From the **Parameter value (disp (m))** list, choose **-5.2**.

3 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Annotation 2

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

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

3 From the **Parameter value (disp (m))** list, choose **-5.2**.

4 Locate the **Position** section. In the **Z** text field, type `a-5.2`.

Surface 3

In the **Model Builder** window, under **Results>Total Displacement, 3D (shell)** right-click **Surface 2** and choose **Duplicate**.

Annotation 3

In the **Model Builder** window, under **Results>Total Displacement, 3D (shell)** right-click **Annotation 2** and choose **Duplicate**.

Surface 3

1 In the **Settings** window for **Surface**, locate the **Data** section.

2 From the **Parameter value (disp (m))** list, choose **-5.8**.

Annotation 3

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

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

3 From the **Parameter value (disp (m))** list, choose **-5.8**.

4 Locate the **Position** section. In the **Z** text field, type `a-5.8`.

5 In the **Total Displacement, 3D (shell)** toolbar, click **Plot**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

In order to better visualize the shell normal in **Thickness and Orientation** plot, reduce the number of arrows.

Shell Local System

1 In the **Model Builder** window, expand the **Thickness and Orientation (shell)** node, then click **Shell Local System**.

2 In the **Settings** window for **Coordinate System Line**, locate the **Positioning** section.

3 In the **Number of arrows** text field, type `20`.

4 In the **Thickness and Orientation (shell)** toolbar, click **Plot**.

Plot a 1D curve showing the relationship between axial displacement and point load at the crown.

1D Plot Group 9

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

2 In the **Settings** window for **1D Plot Group**, type `Load vs. Displacement at Crown` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Load vs. Displacement at Crown`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1 Right-click **Load vs. Displacement at Crown** and choose **Point Graph**.

2 Select Point 1 only.

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

4 In the **Expression** text field, type `Fn1`.

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

6 In the **Expression** text field, type `wn1`.

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
`Solid Mechanics, 40 Elements`

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Shell/Parametric Solutions 1 (sol3)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `Fn2`.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `wn2`.

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

Legends
`Shell, 4 Elements`
`Shell, 8 Elements`
`Shell, 16 Elements`

7 In the **Load vs. Displacement at Crown** toolbar, click **Plot**.

Point Evaluation 1

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

2 In the **Settings** window for **Point Evaluation**, type `Solid Mechanics, 40 Elements` in the **Label** text field.

3 Select Point 1 only.

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

Expression	Unit	Description
`Fn1`	`1`	`Nondimensional force (Solid Mechanics, 40 Elements)`
`wn1`	`1`	`Nondimensional displacement (Solid Mechanics, 40 Elements)`

5 Click **Evaluate**.

Solid Mechanics, 40 Elements 1

1 Right-click **Solid Mechanics, 40 Elements** and choose **Duplicate**.

2 In the **Settings** window for **Point Evaluation**, type `Shell, 4 Elements` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Shell/ Parametric Solutions 1 (sol3)**.

4 From the **Parameter selection (meshdist)** list, choose **From list**.

5 In the **Parameter values (meshdist)** list, select **4**.

6 From the **Table columns** list, choose **Outer solutions**.

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

Expression	Unit	Description
`Fn2`	`1`	`Nondimensional force (Shell, 4 Elements)`
`wn2`	`1`	`Nondimensional displacement (Shell, 4 Elements)`

8 Click **Evaluate**.

Shell, 4 Elements 1

1 Right-click **Shell, 4 Elements** and choose **Duplicate**.

2 In the **Settings** window for **Point Evaluation**, type `Shell, 8 Elements` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (meshdist)** list, select **8**.

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

Expression	Unit	Description
`Fn2`	`1`	`Nondimensional force (Shell, 8 Elements)`
`wn2`	`1`	`Nondimensional displacement (Shell, 8 Elements)`

5 Click **Evaluate**.

Shell, 8 Elements 1

1 Right-click **Shell, 8 Elements** and choose **Duplicate**.

2 In the **Settings** window for **Point Evaluation**, type `Shell, 16 Elements` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (meshdist)** list, select **16**.

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

Expression	Unit	Description
`Fn2`	`1`	`Nondimensional force (Shell, 16 Elements)`
`wn2`	`1`	`Nondimensional displacement (Shell, 16 Elements)`

5 Click **Evaluate**.

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 constrained.

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.

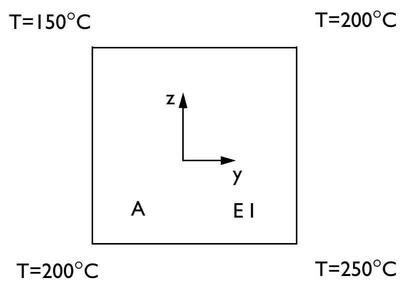


Figure 1: Geometric properties and thermal loads at corners.

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.

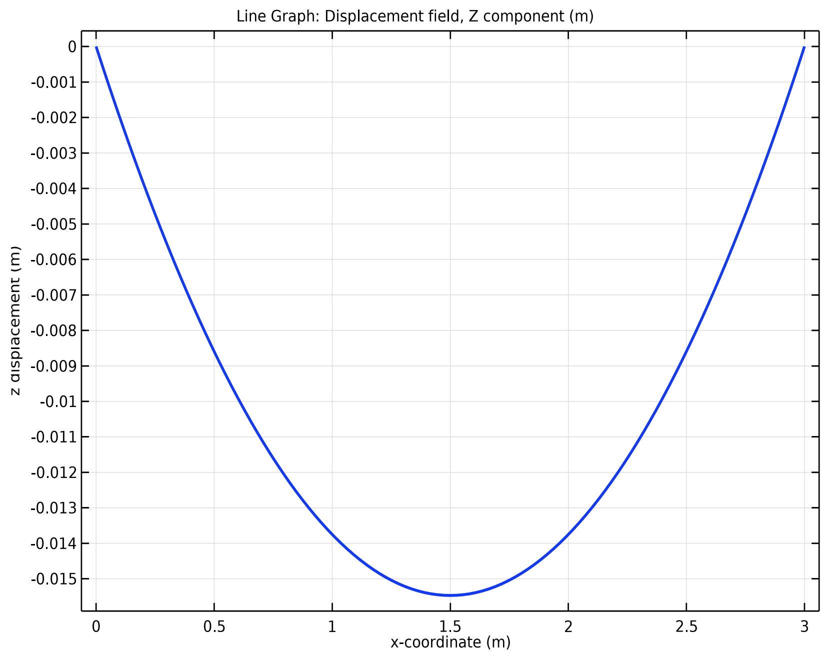


Figure 2: z-displacement along the beam.

The analytical values for the maximum total transverse displacement 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 transverse displacement calculated with COMSOL Multiphysics and the analytical value is shown in the table below.

COMSOL Multiphysics	**Analytical**
21.9 mm	21.9 mm

Figure 3 shows the total displacement, the total transverse displacement and the axial displacement along the beam.

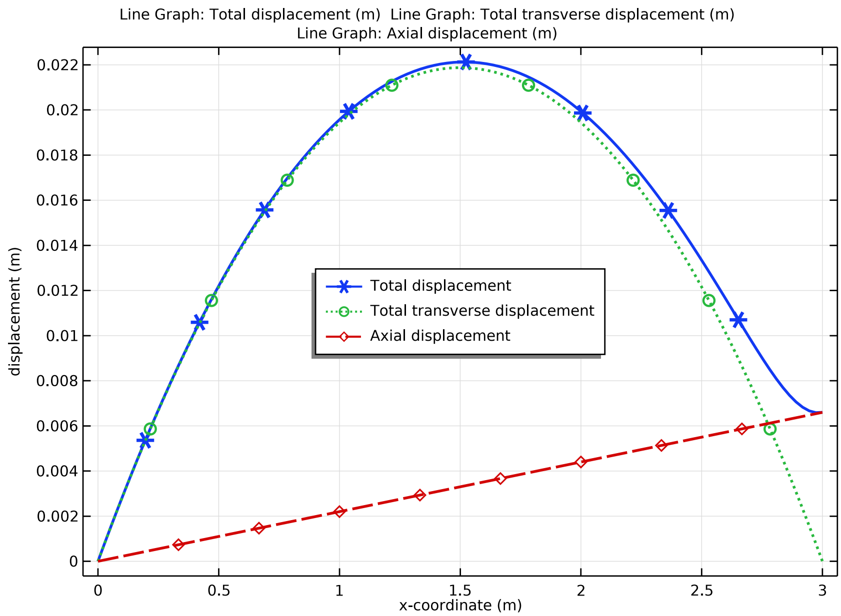


Figure 3: Camber along the beam.

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 **General Studies>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
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
deltaT	50[K]	50 K	Temperature difference
Tg	deltaT/a	1250 K/m	Temperature gradient
Lb	3[m]	3 m	Beam length

GEOMETRY 1

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.
2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.
3 In the table, enter the following settings:

x (m)	y (m)	z (m)
0	0	0
Lb/2	0	0
Lb	0	0

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. 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	Variable	Value	Unit	Property group
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	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, 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 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. In 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. From the list, choose **Rotation**.
8. Select the **Free rotation around y direction** check box.
9. Select the **Free rotation around z direction** check box.

Prescribed Displacement/Rotation 2

1. In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.
2. Select Point 3 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, click **Linear Elastic Material 1**.

Thermal Expansion 1

1. In the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.
2. In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.
3. Click **Go to Source**.

GLOBAL DEFINITIONS

Default Model Inputs

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

2 In the **Settings** window for **Default Model Inputs**, locate the **Browse Model Inputs** section.

3 Find the **Expression for remaining selection** subsection. In the **Volume reference temperature** text field, type `0`.

BEAM (BEAM)

Thermal Expansion 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)>Linear Elastic Material 1** click **Thermal Expansion 1**.

2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.

3 From the T list, choose **User defined**. In the associated text field, type `200`.

4 Locate the **Thermal Bending** section. In the T_{gy} text field, type `Tg`.

5 In the T_{gz} text field, type `-Tg`.

STUDY 1

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

RESULTS

Stress (beam)

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 - m**.

3 In the **Displacements** toolbar, click **Plot**.

1D Plot Group 9

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

2 In the **Settings** window for **1D Plot Group**, type `Transverse Displacement` in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **y-axis label** check box.

4 In the associated text field, type `z displacement (m)`.

Line Graph 1

1 Right-click **Transverse Displacement** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Transverse displacement (z-direction)` in the **Label** text field.

3 Click in the **Graphics** window and then press Ctrl+A to select both edges.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `w`.

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

6 In the **Expression** text field, type `x`.

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

8 In the **Transverse Displacement** toolbar, click **Plot**.

1D Plot Group 10

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

2 In the **Settings** window for **1D Plot Group**, type `Displacement` in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **y-axis label** check box.

4 In the associated text field, type `displacement (m)`.

5 Locate the **Legend** section. From the **Position** list, choose **Center**.

Line Graph 1

1 Right-click **Displacement** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Total displacement` in the **Label** text field.

3 Click in the **Graphics** window and then press Ctrl+A to select both edges.

4 Locate the **y-Axis Data** section. Select the **Description** check box.

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

6 In the **Expression** text field, type `x`.

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

8 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

9 In the **Width** text field, type `2`.

10 Click to expand the **Legends** section. Find the **Include** subsection. Select the **Description** check box.

11 Clear the **Solution** check box.

12 Select the **Show legends** check box.

Total displacement 1

1. Right-click **Total displacement** and choose **Duplicate**.
2. In the **Settings** window for **Line Graph**, type `Total transverse displacement` in the **Label** text field.
3. Locate the **Selection** section. Select the **Activate selection** toggle button.
4. Locate the **y-Axis Data** section. In the **Expression** text field, type `sqrt(v^2+w^2)`.
5. In the **Description** text field, type `Total transverse displacement`.

Total transverse displacement 1

1. Right-click **Total transverse displacement** and choose **Duplicate**.
2. In the **Settings** window for **Line Graph**, type `Axial displacement` in the **Label** text field.
3. Locate the **y-Axis Data** section. In the **Expression** text field, type `u`.
4. In the **Description** text field, type `Axial displacement`.
5. In the **Displacement** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.5

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.

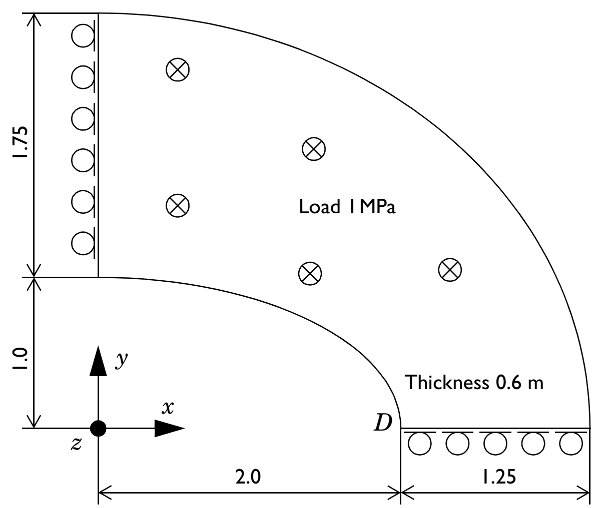


Figure 1: The thick plate geometry, reduced to a quarter of the ellipse due to symmetry.

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.

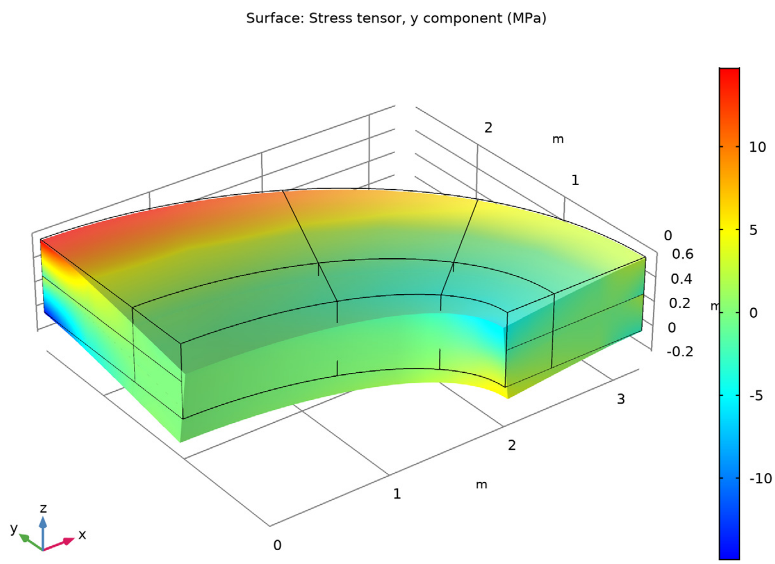


Figure 2: The stress in the y direction.

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 to 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 **General Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

If you do not want to build all the geometry, you can load the geometry sequence from the stored model. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Insert Sequence**. Browse to the model's Application Libraries folder and double-click the file **thick_plate.mph**. You can then continue to the **Add Material** section below.

To build the geometry from a scratch, continue here.

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

Right-click **Work Plane 1 (wp1)** and choose **Show Work Plane**.

Work Plane 1 (wp1)>Ellipse 1 (e1)

1 In the **Work Plane** toolbar, click **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 Click **Build Selected**.
7 Click the **Zoom Extents** button in the **Graphics** toolbar.

Work Plane 1 (wp1)>Ellipse 2 (e2)

1 In the **Work Plane** toolbar, click **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 Click **Build Selected**.

Work Plane 1 (wp1)>Ellipse 3 (e3)

1 In the **Work Plane** toolbar, click **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 Click **Build Selected**.

Work Plane 1 (wp1)>Difference 1 (dif1)

1 In 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 **Activate selection** toggle button.

5 Select the object **e2** only.

6 Click **Build Selected**.

Work Plane 1 (wp1)>Polygon 1 (pol1)

1 In the **Work Plane** toolbar, click **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 table, enter the following settings:

xw (m)	yw (m)
1.783	2.3
1.165	0.812

Work Plane 1 (wp1)>Polygon 2 (pol2)

1 In the **Work Plane** toolbar, click **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 table, enter the following settings:

xw (m)	yw (m)
2.833	1.348
1.783	0.453

5 In the **Work Plane** toolbar, click **Build All**.

Work Plane 1 (wp1)>Plane Geometry

Click the **Zoom Extents** button in the **Graphics** toolbar.

Work Plane 1 (wp1)>Partition Objects 1 (par1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

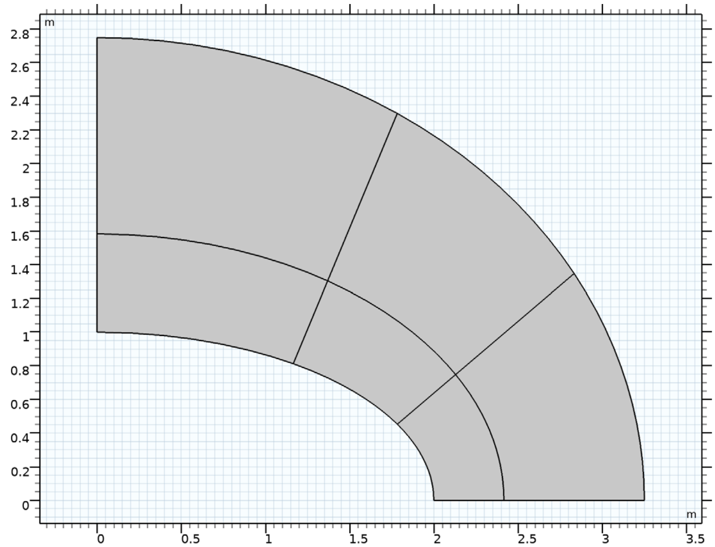
2 Select the object **dif1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 Find the **Tool objects** subsection. Select the **Activate selection** toggle button.

5 Select the objects **pol1** and **pol2** only.

6 Click **Build Selected**.



Work Plane 1 (wp1)

In the **Model Builder** window, click **Work Plane 1 (wp1)**.

Extrude 1 (ext1)

1 In the **Geometry** toolbar, click **Extrude**.

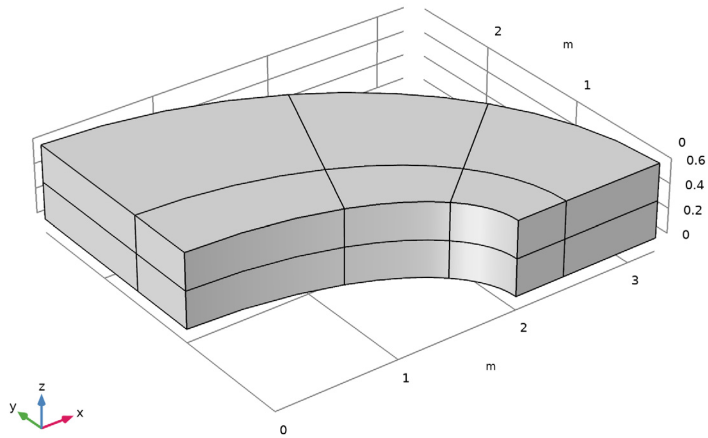
2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
`0.3`
`0.6`

4 Click **Build Selected**.

5 Click the **Zoom Extents** button in 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	Variable	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 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **More Constraints>Symmetry**.

2 Select Boundaries 1, 4, 8, 11, 40, 41, 49, and 50 only.

Prescribed Displacement 1

1. In 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. In 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. In 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

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

Distribution 1

In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

Mapped 1

Select Boundaries 7, 14, 23, 30, 39, and 48 only.

Distribution 1

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

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

RESULTS

Point Evaluation 1

1 In 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) - N/m²>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 (COMSOL)
-5.38[MPa]	MPa	Stress tensor, y compoent, (NAFEMS)

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) - N/m²>solid.sy - Stress tensor, y component**.

3 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

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

Instability of Two Contacting Arches

Introduction

This conceptual example shows how to calculate critical points in models with contact. The model consists of two contacting arches modeled with the Shell interface. During loading, the lower arch exhibits a snap-through behavior. The definition of the problem is based on a benchmark example from Ref. 1.

Model Definition

The model geometry consists of an arch and a block as shown in Figure 1. Since the arches are modeled with the Shell interface, a 3D geometry is used. However, a 2D plane strain behavior is intended, and consequently symmetry conditions are applied to all edges in the *y* direction to suppress any out-of-plane deformation

.

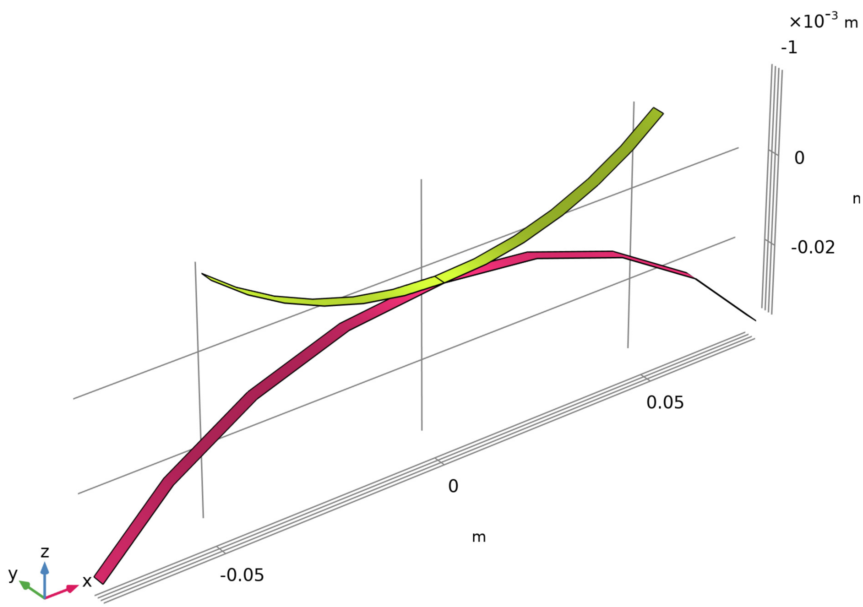


Figure 1: Model geometry.

Only contact without friction is considered and the penalty contact method is used.

The ends of the upper arch are constrained against displacement in the *x* direction and subjected to vertical edge loads. The magnitude of the edge loads is controlled by the monotonically increasing deflection of the upper arch, which makes it possible to track the entire load path, even though the force does not increase monotonically. The ends of the lower arch are fixed.

Results and Discussion

Figure 2 depicts the deformed shape and the von Mises stress distribution at the last step of the simulation. The snap-through of the lower arch is clearly visible. Both arches are represented by a shell dataset that shows both their top and bottom surfaces.

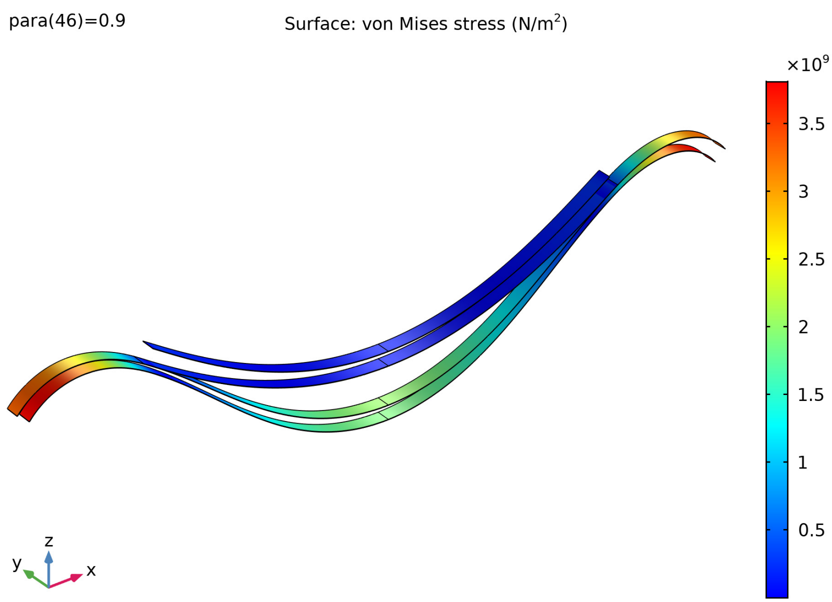


Figure 2: Deformation and von Mises stress at the final step.

Three different load versus deflection curves are shown in Figure 3. The load is represented by a dimensionless load factor, and is plotted against either the mid deflections of the two arches or the average deflection of the ends of the upper arch. Several critical points can be observed. For example, looking at the lower arch, a first limit point is reached for a load factor equal to 107.5 and a deflection of 13 mm. At this point the lower arch becomes unstable and a snap-through occurs. When the deflection reaches 45 mm, the load factor has decreased to 45. At this point a second limit point is reached, and the model finds a new stable configuration. After this point the load factor increases with increasing deflection.

Several bifurcation points are also present, indicating the unstable nature of the problem and possible branching of the load path. A first point is, for example, visible already at a deflection of 1 mm, where there is a clear change in the slope of the load-deflection curve.

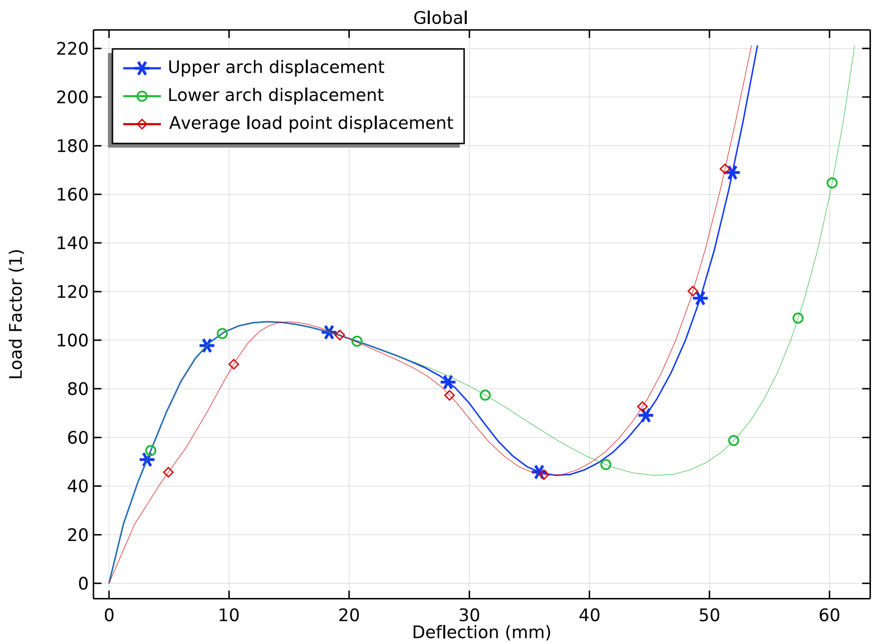


Figure 3: Load versus deflection curves.

The progressive deformation of the two arches, including the snap-through of the lower arch, is shown in Figure 4 for five values of the continuation parameter. In the figure, it also is clearly visible how the contact problem changes throughout the simulation. Figure 5 shows the contact pressure exerted by the upper arch on the lower arch during the post-critical stage.

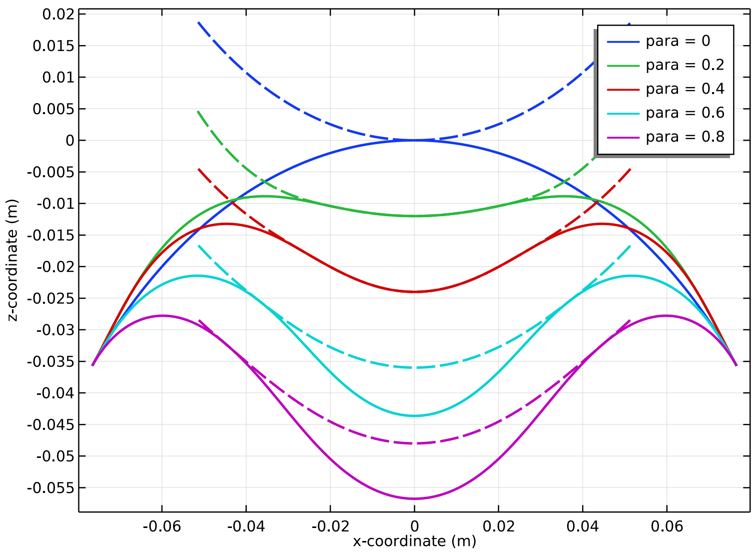


Figure 4: Deformation of the model for five different parameter values

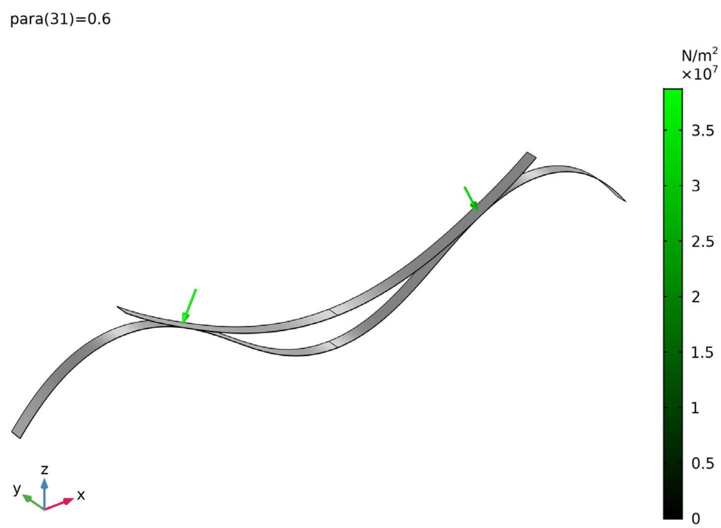


Figure 5: Contact pressure acting on the lower arch.

Notes About the COMSOL Implementation

Contact problems are often unstable in their initial configuration. To help the solver find an initial solution, a **Spring Foundation** is added to the otherwise unconstrained upper arch during the first parameter step.

Modeling the post-critical behavior of a system is not possible by incrementally increasing the boundary load. The unstable behavior is even more pronounced when contact is present. To be able to find all limit points and to track the full load versus deflection curve, a displacement controlled load scheme is used by adding a **Global Equation**. Here, the magnitude of the edge loads is controlled through the monotonically increasing deflection of the upper arch. Alternatively, the vertical displacement could be prescribed on end points of the upper arch, but this is a less general technique that fails for some cases.

This problem is highly unstable and several branches of the equilibrium path are possible. To suppress these so that a stable solution is obtained, the mid-point of both arches is constrained against sideways displacement through a symmetry condition. By deactivating this constraint, it is possible to study the branching of the equilibrium path.

Reference

1. P. Wriggers, *Computational Contact Mechanics*, Springer-Verlag, 2006

Application Library path: `Structural_Mechanics_Module/Verification_Examples/two_arches`

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 **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `two_arches_parameters.txt`.

GEOMETRY 1

Work Plane 1 (wp1)

1 In 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**.

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `Ri_upper`.

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

6 Locate the **Position** section. In the **yw** text field, type `Ri_upper`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90-seg_upper/2`.

8 Click **Build Selected**.

9 Click the **Zoom Extents** button in the **Graphics** toolbar.

Work Plane 1 (wp1)>Circle 2 (c2)

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `Ri_lower`.

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

6 Locate the **Position** section. In the **yw** text field, type `-Ri_lower`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `90-seg_lower/2`.

8 Click **Build Selected**.

9 Click the **Zoom Extents** button in the **Graphics** toolbar.

Work Plane 1 (wp1)>Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Plane Geometry** and choose **Delete Entities**.

2 On the object **c1**, select Boundaries 2 and 3 only.

3 On the object **c2**, select Boundaries 3 and 4 only.

Work Plane 1 (wp1)>Partition Edges 1 (pare1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Partition Edges**.

2 On the object **del1(1)**, select Boundary 1 only.

Work Plane 1 (wp1)

1 In the **Model Builder** window, click **Work Plane 1 (wp1)**.

2 In the **Settings** window for **Work Plane**, locate the **Unite Objects** section.

3 Clear the **Unite objects** check box.

Extrude 1 (ext1)

1 In 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)
d

4 Click **Build Selected**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Explicit Selection 1 (sel1)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Upper Arch` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Object**.

4 Select the object **ext1(2)** only.

5 Locate the **Color** section. From the **Color** list, choose **Color 4**.

6 Click **Build Selected**.

Upper Arch 1 (sel2)

1 Right-click **Upper Arch** and choose **Duplicate**.

2 In the **Settings** window for **Explicit Selection**, type `Lower Arch` in the **Label** text field.

3 Locate the **Entities to Select** section. In the list, select **ext1(2)**.

4 Select the object **ext1(1)** only.

5 Locate the **Color** section. From the **Color** list, choose **Color 12**.

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 Click **Build Selected**.

5 Click the **Zoom Extents** button in 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 **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Lower Arch**.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	40[GPa]	Pa	Basic
Poisson's ratio	nu	0.2	1	Basic
Density	rho	1	kg/m³	Basic

Material 2 (mat2)

1 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 **Upper Arch**.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	20[GPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	1	kg/m³	Basic

DEFINITIONS

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

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

4 Select Point 9 only.

Average 2 (aveop2)

1 Right-click **Average 1 (aveop1)** and choose **Duplicate**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

3 Click **Clear Selection**.

4 Select Point 3 only.

Average 3 (aveop3)

1 Right-click **Average 2 (aveop2)** and choose **Duplicate**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

3 Click **Clear Selection**.

4 Select Points 7 and 11 only.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
disp_upper	aveop1(-w)	m	Upper arch displacement
disp_lower	aveop2(-w)	m	Lower arch displacement
disp_load	aveop3(-w)	m	Average load point displacement

Contact Pair 1 (p1)

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

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

3 From the **Selection** list, choose **Upper Arch**.

4 Locate the **Destination Boundaries** section. From the **Selection** list, choose **Lower Arch**.

SHELL (SHELL)

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

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

4 From the **Offset definition** list, choose **Relative offset**.

5 In the $z_{\mathrm{reloffset}}$ text field, type `-1`.

Symmetry 1

1 In the **Physics** toolbar, click **Edges** and choose **Symmetry**.

2 Select Edges 2, 3, 5, 6, 9, 10, 12, and 13 only.

Prescribed Displacement/Rotation 1

1 In the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement/Rotation**.

2 Select Edges 8 and 14 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.

Prescribed Displacement/Rotation 2

1 In the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement/Rotation**.

2 Select Edges 1 and 7 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. From the **By** list, choose **Rotation**.

Contact 1

1 In 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 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (p1)** in the **Pairs** list.

5 Click **OK**.

Edge Load 1

1 In the **Physics** toolbar, click **Edges** and choose **Edge Load**.

2 Select Edges 8 and 14 only.

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

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

0	x
0	y
load*F_ref	z

The dependent variable `load` will be created in the next step using a global equation.

5 Click the **Show More Options** button in the **Model Builder** toolbar.

6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

7 Click **OK**.

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **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
load	disp_up per-max_disp*para	0	0	Load Factor

4 Locate the **Units** section. Click **Select Source Term Quantity**.

5 In the **Physical Quantity** dialog box, type `displacement` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Displacement (m)**.

8 Click **OK**.

Add a small spring stiffness to the upper arch to stabilize the model during the initial step.

Spring Foundation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.

2 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

3 In the $\mathbf{k}_A$ text field, type `1e3*(para<0.01)`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Upper Arch**.

Several possible branches are possible during the snap-through. Adding a constraint to each arch enforces a symmetric and stable solution.

Symmetry 2

1 In the **Physics** toolbar, click **Edges** and choose **Symmetry**.

2 Select Edges 4 and 11 only.

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 **Mapped 1** and choose **Distribution**.

2 Select Edges 2 and 5 only.

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

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

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 9 and 12 only.

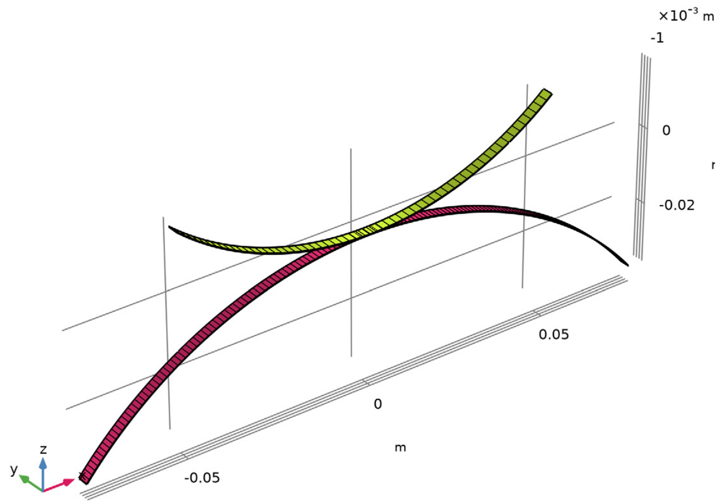
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

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

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

6 In the **Settings** window for **Mesh**, click **Build All**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar.



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 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 (Load parameter)	`range(0,0.02,1)`	

Solution 1 (sol1)

In the **Study** toolbar, click **Show Default Solver**.

Stationary Solver 1

1 In the **Model Builder** window, expand the **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 `0.0005`.

Parametric 1

1 In the **Model Builder** window, expand the **Stationary Solver 1** node, then click **Parametric 1**.

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

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

4 In the **Minimum step size** text field, type `1e-6`.

Stop Condition 1

1 Right-click **Parametric 1** and choose **Stop Condition**.

2 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.

3 Click **Add**.

4 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
`comp1.load/250`	True (>=1)	√	`Stop expression 1`

5 Locate the **Output at Stop** section. From the **Add solution** list, choose **Step before stop**.

Fully Coupled 1

1 In the **Model Builder** window, click **Fully Coupled 1**.

2 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

3 From the **Nonlinear method** list, choose **Constant (Newton)**.

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

RESULTS

Stress (shell)

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

2 From the **Frame** list, choose **Spatial (x, y, z)**.

3 In the **Stress (shell)** toolbar, click **Plot**.

4 Click the **Show Grid** button in the **Graphics** toolbar.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Contact (shell)

1 In the **Model Builder** window, click **Contact (shell)**.

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

3 From the **Parameter value (para)** list, choose **0.6**.

Contact 1, Pressure

1 In the **Model Builder** window, expand the **Contact (shell)** node, then click **Contact 1, Pressure**.

2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.

3 Select the **Scale factor** check box.

4 In the associated text field, type `2e-10`.

Gray Surfaces

In the **Model Builder** window, right-click **Gray Surfaces** and choose **Enable**.

Animation 1

1 In the **Contact (shell)** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Frames** section.

3 From the **Frame selection** list, choose **All**.

4 Right-click **Animation 1** and choose **Play**.

1D Plot Group 6

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

2 In the **Settings** window for **1D Plot Group**, type `Load vs Deflection` in the **Label** text field.

Global 1

1 Right-click **Load vs Deflection** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`disp_upper`	`mm`	`Upper arch displacement`
`disp_lower`	`mm`	`Lower arch displacement`
`disp_load`	`mm`	`Average load point displacement`

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

5 In the **Expression** text field, type `load`.

6 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

Load vs Deflection

1 In the **Model Builder** window, click **Load vs Deflection**.

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

3 Select the **Flip the x- and y-axes** check box.

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Deflection (mm)`.

7 In the **Load vs Deflection** toolbar, click **Plot**.

1D Plot Group 7

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

2 In the **Settings** window for **1D Plot Group**, type `Deformation` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (para)** list, choose **Manual**.

4 In the **Parameter indices (1-46)** text field, type `range(1,10,41)`.

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

Line Graph 1

1 Right-click **Deformation** and choose **Line Graph**.

2 Select Edges 2 and 5 only.

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

4 In the **Expression** text field, type `z`.

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

6 In the **Expression** text field, type `x`.

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

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 Click **Clear Selection**.

4 Select Edges 9 and 12 only.

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

6 From the **Color** list, choose **Cycle (reset)**.

Line Graph 1

1 In the **Model Builder** window, click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 Find the **Include** subsection. In the **Prefix** text field, type `para = `.

5 In the **Deformation** toolbar, click **Plot**.

Stress (shell)

Click the **Zoom Extents** button in the **Graphics** toolbar.

Created in COMSOL Multiphysics 5.5

Heat Generation in a Vibrating Structure

Introduction

When a structure is subjected to vibrations of high frequency, a significant amount of heat can be generated within the structure because of mechanical losses in the material such as, for example, viscoelastic effects.

In this example, you model the slow rise of the temperature in a vibrating beam-like structure. You use a transient heat-transfer problem with source term which represents the heat generation due to mechanical losses. The simulation is based on a structural analysis performed in the frequency domain.

Model Definition

The beam consists of two layers made of aluminum and titanium, respectively, with the corresponding loss factors 0.001 and 0.005. One end of the beam is fixed, and the other one is subjected to periodic loading in the z direction, which is represented in the frequency domain as $F_z \exp(j\omega t)$, where j is the imaginary unit, and the angular frequency is

$$\omega = 2\pi f$$

The excitation frequency f = 7767 Hz and the load magnitude F_z = 1.7 MPa are used in this example.

The temperature rise is given by the heat-transfer equation

$$\rho C_p \frac{\partial T}{\partial t} - \nabla \cdot (k \nabla T) = Q_h$$

where k is the thermal conductivity, and the volumetric heat capacity ρC_p is independent of the temperature in accordance with the Dulong-Petit law.

Note that T represents the temperature averaged over the time period $2\pi/\omega$. The heat source

$$Q_h = \frac{1}{2}\omega\eta \mathrm{Real}[\varepsilon : \mathrm{Conj}(\mathsf{C} : \varepsilon)]$$

presents the internal work of the nonelastic (for example, viscous) forces over the period. In the above expression, η is the loss factor, ε is the strain tensor, and C is the elasticity tensor. The term is computed from a structural analysis performed in the frequency domain.

The initial state at time $t = 0$ is stress-free, and the initial temperature is 293.15 K over the entire beam.

Use the following boundary conditions:

- At the fixed end, use the temperature condition $T = 293.15$ K.
- At the end subjected to periodic force, use the thermal insulation condition.
- The boundary between the layers of different materials is an interior boundary.
- At all other boundaries, use the convective cooling condition:

$$\mathbf{n} \cdot (-k\nabla T) = h(T - T_{\mathrm{ext}})$$

where $h = 5$ W/(m^2·K) is the heat transfer coefficient and $T_{\mathrm{ext}} = 293.15$ K is the external temperature.

For the simulation, apply a periodic loading in the z direction of magnitude 1.7 MPa and frequency 7767 Hz at the free end of the beam for 2 seconds, keeping the fixed end and the structure environment at a constant temperature of 300 K during the process.

Results and Discussion

Figure 1 displays the temperature distribution at the end of the simulated 2-second forced vibrations. As the figure shows, the maximum temperature rise in the beam is about 0.2 K.

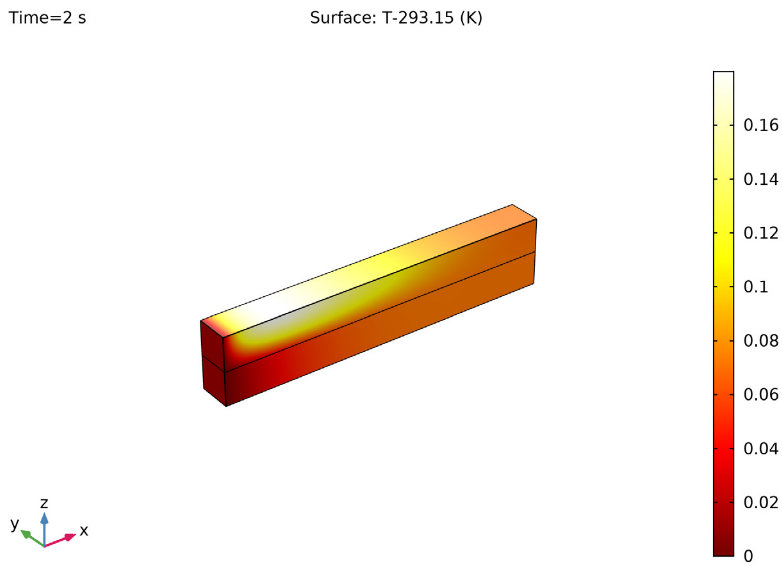


Figure 1: Temperature increase in the beam after 2 seconds of forced vibrations.

Application Library path: `Structural_Mechanics_Module/Thermal-Structure_Interaction/vibrating_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 **Heat Transfer>Heat Transfer in Solids (ht)**.

3 Click **Add**.

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

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Time Dependent**.

8 Click **Done**.

GEOMETRY 1

Block 1 (blk1)

1 In 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 `0.01`.

4 In the **Depth** text field, type `0.001`.

5 In the **Height** text field, type `0.001`.

Block 2 (blk2)

1 In 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 `0.01`.

4 In the **Depth** text field, type `0.001`.

5 In the **Height** text field, type `0.001`.

6 Locate the **Position** section. In the **z** text field, type `0.001`.

7 In the **Model Builder** window, click **Geometry 1**.

8 Click **Build All Objects**.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in>Aluminum**.

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

5 In the tree, select **Built-in>Titanium beta-21S**.

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

7 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Aluminum (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat1)**.

2 Select Domain 1 only.

Titanium beta-21S (mat2)

1 In the **Model Builder** window, click **Titanium beta-21S (mat2)**.

2 Select Domain 2 only.

SOLID MECHANICS (SOLID)

You need to set up the Solid Mechanics equation form to frequency-domain, since the study type will be set to time dependent. The time dependent equations should be applied to the heat transfer physics only.

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

2 In the **Settings** window for **Solid Mechanics**, click to expand the **Equation** section.

3 From the **Equation form** list, choose **Frequency domain**.

4 From the **Frequency** list, choose **User defined**. In the f text field, type `7767`.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundaries 1 and 4 only.

Boundary Load 1

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

2 Select Boundaries 10 and 11 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
1.7[MPa]	z

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Damping 1

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.

2 Select Domain 1 only.

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

4 From the **Damping type** list, choose **Isotropic loss factor**.

5 From the η_s list, choose **User defined**. In the associated text field, type `0.001`.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Damping 2

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.

2 Select Domain 2 only.

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

4 From the **Damping type** list, choose **Isotropic loss factor**.

5 From the η_s list, choose **User defined**. In the associated text field, type `0.005`.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Temperature 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundaries 1 and 4 only.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

3 Click the **Convective heat flux** button.

4 In the h text field, type `5`.

5 Select Boundaries 2, 3, 5, and 7–9 only.

Heat Source 1

1 In the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 In the **Settings** window for **Heat Source**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Heat Source** section. From the Q_0 list, choose **Total power dissipation density (solid/lemm1)**.

This choice models the heat generated by the vibrations in the structure.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Extra fine**.

Swept 1

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Swept**.

2 Click **Build All**.

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 `range(0,0.05,2)`.

Before computing the solution, generate the default plots.

4 In the **Model Builder** window, right-click **Study 1** and choose **Get Initial Value for Step**.

RESULTS

Surface

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

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

3 In the **Expression** text field, type `T-293.15`.

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**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

Solver Configurations

In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

Solution 1 (sol1)

You need to enable complex values because they are used in the solid mechanics equations, which you manually reconfigured for the frequency-domain analysis.

1. In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.
2. In the **Settings** window for **Time-Dependent Solver**, click to expand the **Advanced** section.
3. Select the **Allow complex numbers** check box.
4. In the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

1. Click the **Zoom Out** button in the **Graphics** toolbar.

   The computed solution should closely resemble that shown in Figure 1.

Created in COMSOL Multiphysics 5.5

Vibrating Membrane

Introduction

In the following example you compute the natural frequencies of a pretensioned 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, ν = 0.33
- Mass density, ρ = 7850 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 pretensioned 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	369.0
5	k_{12} = 5.1356	369.0	369.0
6	k_{20} = 5.5201	396.6	396.7

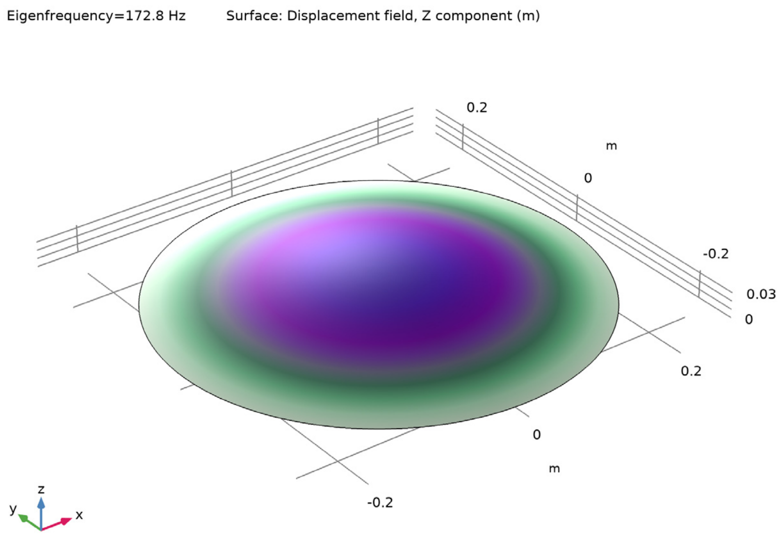


Figure 1: First eigenmode.

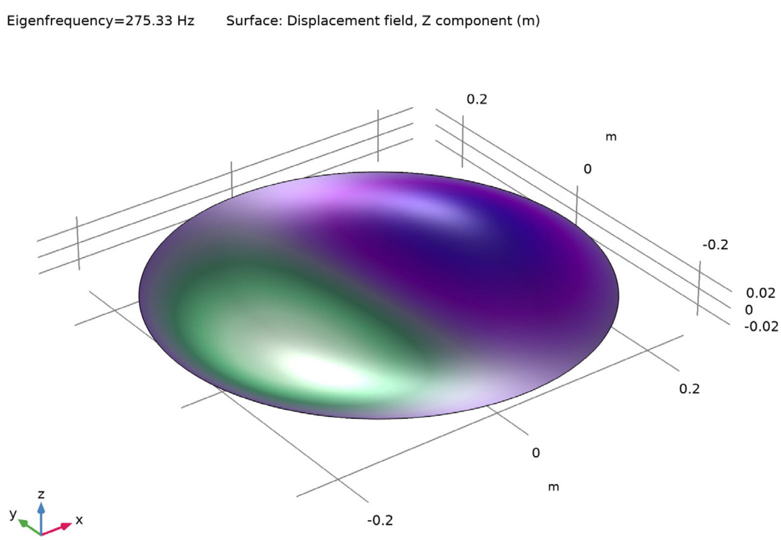


Figure 2: Second eigenmode.

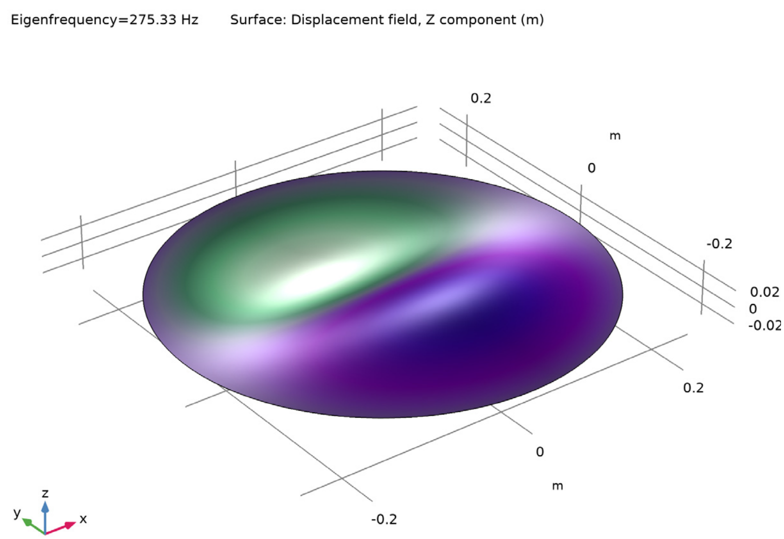


Figure 3: Third eigenmode.

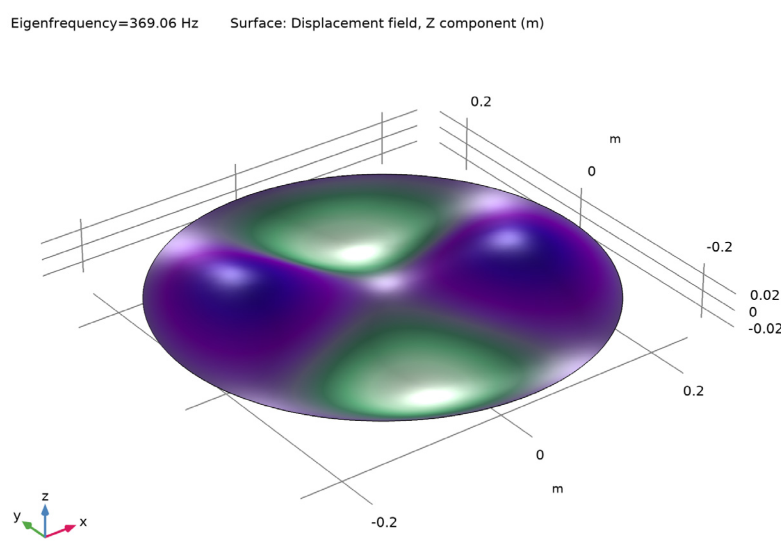


Figure 4: Fourth eigenmode.

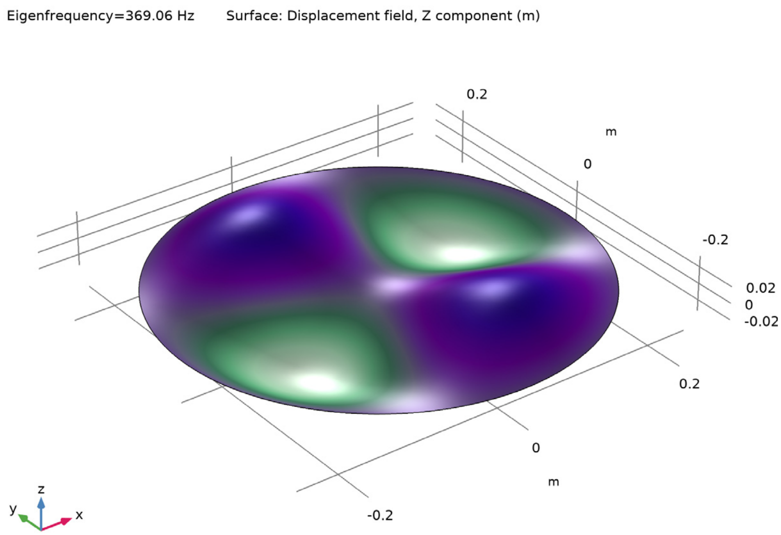


Figure 5: Fifth eigenmode.

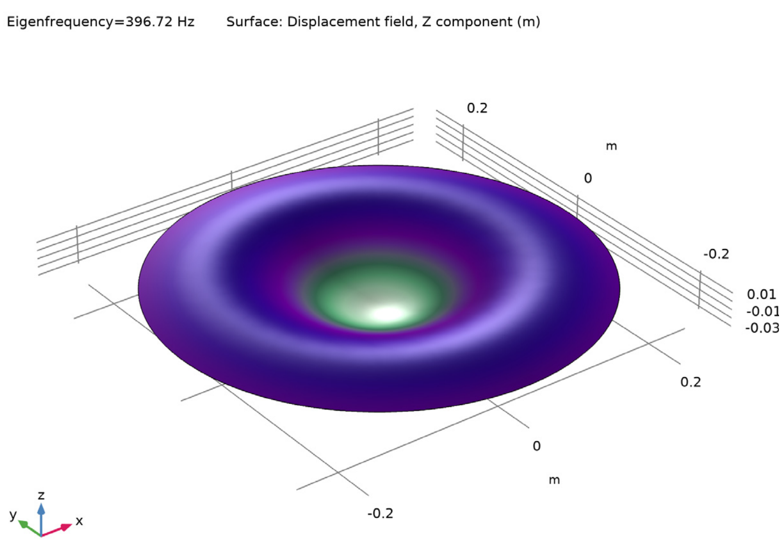


Figure 6: Sixth eigenmode.

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 **General Studies>Eigenfrequency**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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	20000 N/m	Pretension 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)	71.853 1/s	Common factor in natural frequencies
f10	2.4048*fct	172.79 1/s	1st natural frequency
f11	3.8317*fct	275.32 1/s	2nd and 3d natural frequencies
f12	5.1356*fct	369.01 1/s	4th and 5th natural frequencies
f20	5.5201*fct	396.64 1/s	6th natural frequency

DEFINITIONS

Cylindrical System 2 (sys2)

In the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

GEOMETRY 1

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

Right-click **Work Plane 1 (wp1)** and choose **Show Work Plane**.

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **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 In the **Home** toolbar, click **Build All**.

6 Click the **Zoom Extents** button in 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	Variable	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)

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Membrane (mbrn)** click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

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

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Initial Stress and Strain 1

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

Prescribed Displacement 2

1 In 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 **Physics-Controlled Mesh** 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 In the **Home** toolbar, click **Compute**.

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Mode Shape (mbrn)** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `w`.

4 In the **Mode Shape (mbrn)** toolbar, click **Plot**.
5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Mode Shape (mbrn)

1 In the **Model Builder** window, click **Mode Shape (mbrn)**.
2 From the **Eigenfrequency** list, choose the first frequency at **275.3** Hz.
3 In the **Mode Shape (mbrn)** toolbar, click **Plot**.
4 From the **Eigenfrequency** list, choose the first frequency at **275.3** Hz.
5 In the **Mode Shape (mbrn)** toolbar, click **Plot**.
6 From the **Eigenfrequency** list, choose the first frequency at **369.1** Hz.
7 In the **Mode Shape (mbrn)** toolbar, click **Plot**.
8 From the **Eigenfrequency** list, choose the first frequency at **369.1** Hz.
9 In 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 **396.72**.
12 In 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 In 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 for Selected Physics Interfaces>Eigenfrequency, Prestressed**.
4 Click **Add Study** in the window toolbar.
5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

MEMBRANE (MBRN)

Edge Load 1

1 In 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. Specify the $\mathbf{F}_L$ 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. In the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Spring Foundation**, locate the **Spring** section.
4. From the list, choose **Diagonal**.
5. In the $\mathbf{k}_A$ 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 model configuration 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, click **Step 2: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.
3 Select the **Modify model configuration for study step** check box.
4 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**.
5 Click **Disable**.
6 In 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

Solver Configurations

1 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.
2 Select the **Modify model configuration 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 5.5

Vibrating String

Introduction

In the following example you compute the natural frequencies of a pretensioned 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 pretension 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 pretensioned 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 pretensioning 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

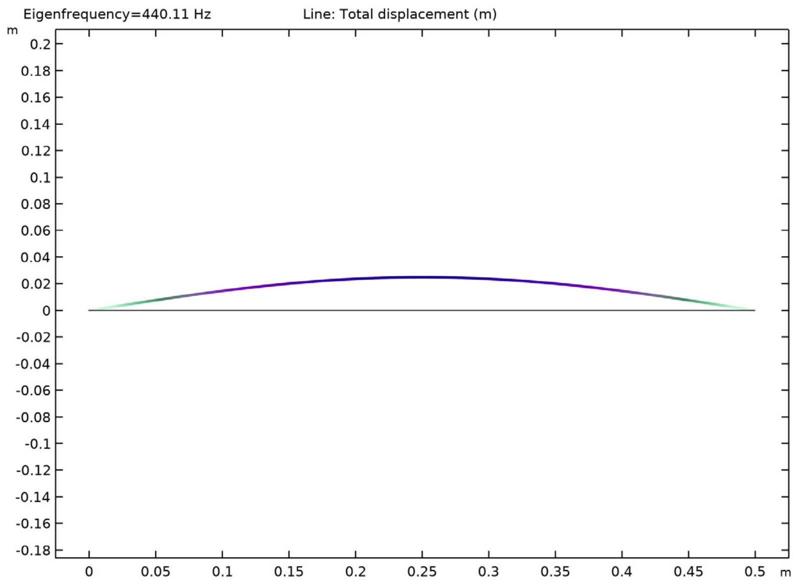


Figure 1: First eigenmode.

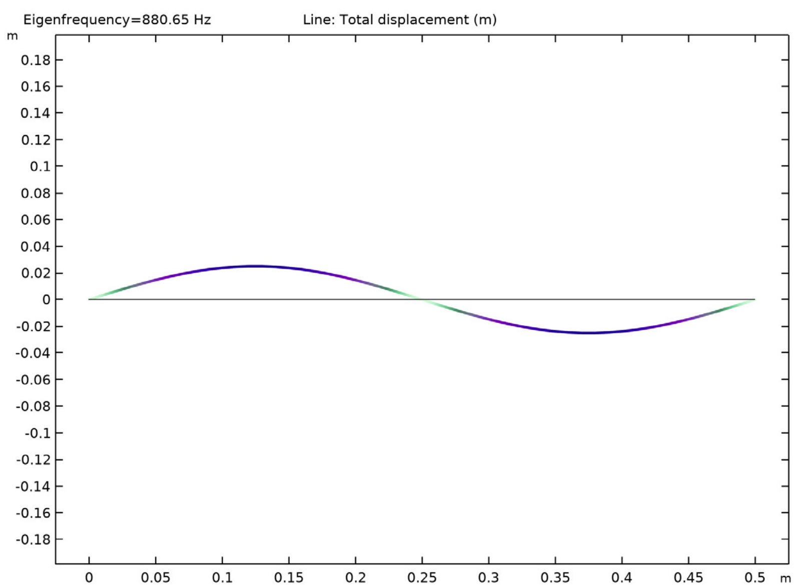


Figure 2: Second eigenmode.

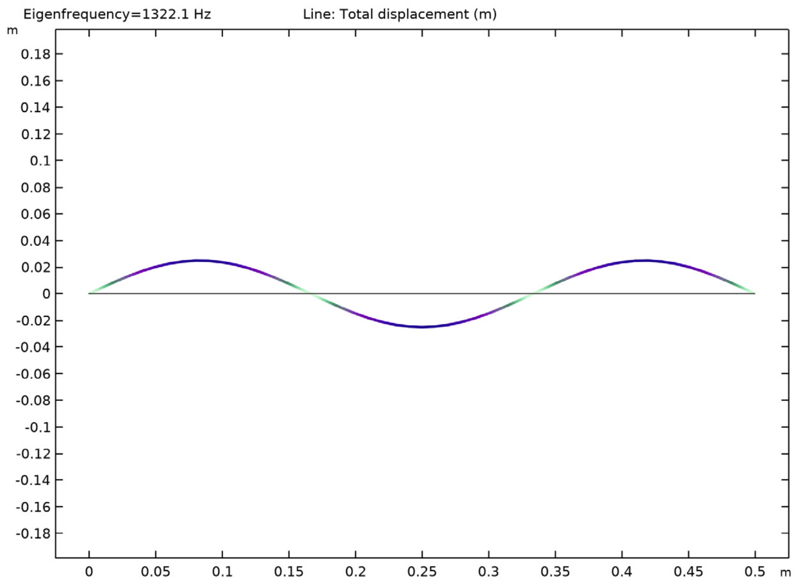


Figure 3: Third eigenmode.

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 string 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 **General Studies>Eigenfrequency**.

6 Click **Done**.

GEOMETRY 1

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

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

3 In the table, enter the following settings:

x (m)	y (m)
0	0
0.5	0

4 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	Variable	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, 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*0.001^2`.

Pinned 1

1. In the **Physics** toolbar, click **Points** 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.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Initial Stress and Strain 1

1. In 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 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, 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 In the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (truss)

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

   The default plot shows the displacement for the first eigenmode.

Line 1

1 In the **Model Builder** window, expand the **Mode Shape (truss)** node, then click **Line 1**.

2 In the **Settings** window for **Line**, locate the **Coloring and Style** section.

3 In the **Radius scale factor** text field, type `2`.

Mode Shape (truss)

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

2 In the **Model Builder** window, click **Mode Shape (truss)**.

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

4 From the **Eigenfrequency (Hz)** list, choose **880.65**.

   This corresponds to the second eigenmode.

5 In the **Mode Shape (truss)** toolbar, click **Plot**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

7 From the **Eigenfrequency (Hz)** list, choose **1322.1**.

   This is the third eigenmode.

8 In the **Mode Shape (truss)** toolbar, click **Plot**.

9 Click the **Zoom Extents** button in 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 **Physics** toolbar, click **Points** and choose **Pinned**.

2 Select Point 1 only.

Prescribed Displacement 1

1 In 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.

Point Load 1

1 In 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

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 In the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

4 From the list, choose **Diagonal**.

5 In the $\mathbf{k}_\mathrm{L}$ table, enter the following settings:

0	0
0	10

ADD STUDY

1 In 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 for Selected Physics Interfaces>Eigenfrequency, Prestressed**.

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

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

2 Select the **Modify model configuration for study step** check box.

3 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**.

4 Click **Disable**.

Step 2: Eigenfrequency

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

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

3 Select the **Modify model configuration 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 In 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.

Line 1

1 In the **Model Builder** window, expand the **Mode Shape (truss) 1** node, then click **Line 1**.

2 In the **Settings** window for **Line**, locate the **Coloring and Style** section.

3 In the **Radius scale factor** text field, type 2.

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, under **Study 1** click **Step 1: Eigenfrequency**.

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

3 Select the **Modify model configuration 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**.