Created in COMSOL Multiphysics 6.0

Various Analyses of an Elbow Bracket

Introduction

The component shown in Figure 1 is part of a support mechanism and is subjected to various loads. This tutorial model takes you through the steps to carry out a detailed analysis of the part using the Structural Mechanics Module.

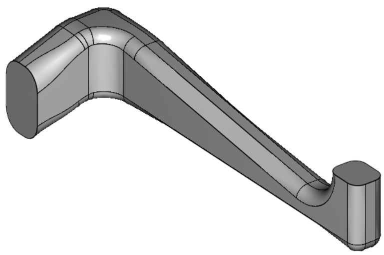


Figure 1: Geometry of the elbow bracket.

In the various parts of the example you are introduced to using some basic analysis types, together with numerous postprocessing possibilities. These analysis types are:

- Static analysis
- Eigenfrequency analysis
- Damped eigenfrequency analysis
- Transient analysis
- Modal based transient analysis
- Frequency response analysis
- Modal based frequency response analysis
- Parametric analysis
- Linear buckling analysis

This tutorial model comes in two versions:

- A short version, `elbow_bracket_brief`, treating the three first analysis types in the above list.
- A complete version, `elbow_bracket`, treating all nine analysis types.

This tutorial model consists of a single model, with nine studies, corresponding to these analysis types, which are described in the section *Study Types* in the *Structural Mechanics Module User's Guide*. The chapter *Structural Mechanics Modeling* in the same manual provides further assistance.

Note: If you have already built the short model version described in the section Static and Eigenfrequency Analyses of an Elbow Bracket, you can proceed directly to the section Time-Dependent Analysis.

Model Definition

The geometry for this part, see Figure 1, has been created with a CAD software, and it is available for you to import into COMSOL Multiphysics.

Material

The material is structural steel, as taken from the material library, with Young's modulus 200 GPa and Poisson's ratio of 0.33.

Damping

The Structural Mechanics Module supports several types damping for dynamic analysis. You can also use no damping, which is the default option.

In some of the studies Rayleigh damping is used. It is defined by two scalar damping parameters that are multipliers to the mass matrix (α_{dM}) and stiffness matrix (β_{dK}) in the following way:

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

where C is the damping matrix, M is the mass matrix, and K is the stiffness matrix. The damping is specified locally in each domain; this means that you can specify different damping parameters in different parts of the model. This is an extension of the common definition of Rayleigh damping.

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

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

where ξ_i is the critical damping ratio at a specific angular frequency ω_i. Knowing two pairs of corresponding ξ_i and ω_i results in a system of equations from which the damping parameters can be determined. This method of determining the Rayleigh damping parameters is built-in.

Assume that the structure has a constant damping ratio of 0.1. Select two frequencies near the excitation frequency, 400 Hz and 600 Hz. This will result in α_{dM} = 302 s^{-1} and β_{dK} = $3.18 \cdot 10^{-5}$ s.

For more information see the section *Mechanical Damping and Losses* in the *Structural Mechanics Module User's Guide*.

If modal based dynamic response studies are performed it is usually easier to give the critical damping ratios directly. This also gives more detailed control over the damping properties over a large frequency range.

Loads and Constraints

The displacement are fixed in all directions on the face shown in Figure 2. The load is described under each study, but in all cases it is distributed over the face as shown in this figure.

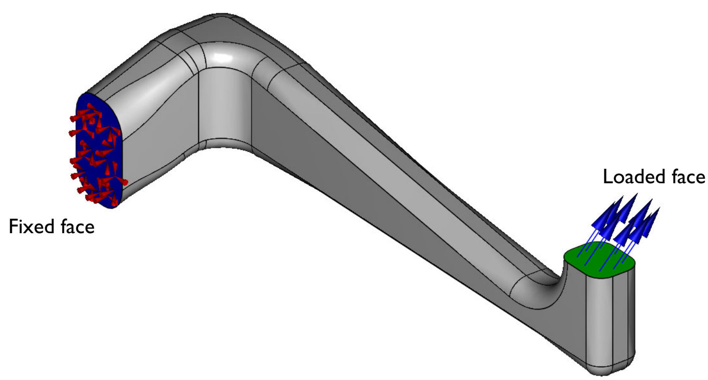


Figure 2: Constraint and loading of the bracket.

The Application Libraries note immediately below appears in the discussion of every model. The path indicates the location of the example file in the Application Libraries root directory. The most convenient way to open it is from the **Application Libraries** window in the COMSOL Desktop, which you can open from the **File** menu.

Application Library path: `Structural_Mechanics_Module/Tutorials/elbow_bracket`

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

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 From the **Source** list, choose **COMSOL Multiphysics file**.

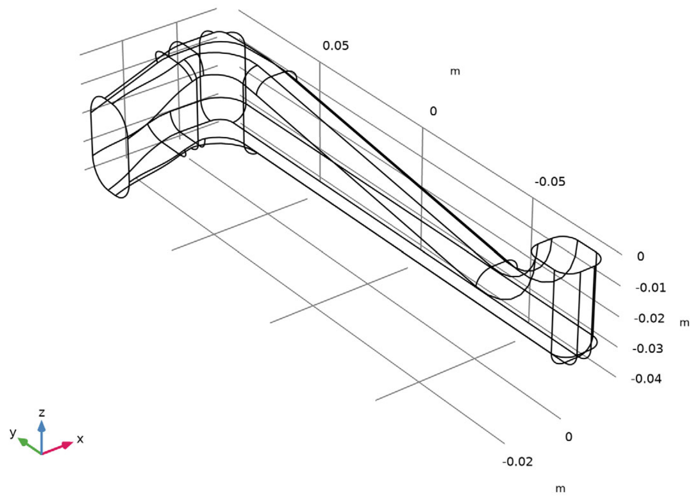
4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `elbow_bracket.mphbin`.

6 Click **Import**.

7 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

The view in the **Graphics** window should look like that in the image below.



8 Click the **Wireframe Rendering** button in the **Graphics** toolbar to return to the default surface rendering.

Suppress some edges during meshing, in order to avoid generation of unnecessary small elements.

Ignore Edges 1 (ige1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **fin**, select Edges 17, 21, 23, 27, 38, 40, 42, and 44 only.

It might be easier to select the correct edges 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.)

MESH 1

Free Tetrahedral 1

In the **Mesh** toolbar, click **Free Tetrahedral**.

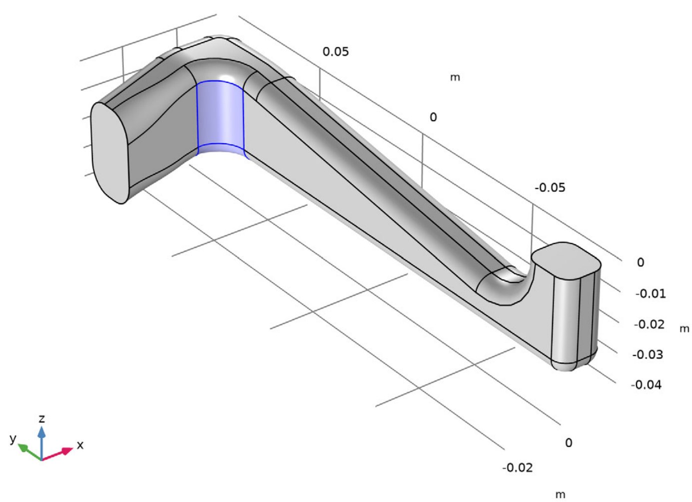
Size

There are nine predefined combinations of mesh parameter settings. They range from **Extremely fine** to **Extremely coarse**, with **Normal** as the default setting. Unless any other mesh parameters are set, this is the setting that is used if you use **Build All** or **Build Selected** to generate the mesh.

As a stress concentration can be expected in the corner of the bracket, put a finer mesh there.

Size 1

1 In the **Model Builder** window, right-click **Free Tetrahedral 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 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.
5 Select Boundaries 13 and 14 only.



6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

MATERIALS

Next, specify the material properties. You can do this either by explicitly typing them in or by selecting a library material in the Material Browser. For this model, use a library material.

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>Structural steel**.

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

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

Static Analysis

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

The purpose of such analysis can be to find the maximum stress level and compare it with the material's yield strength, as well as to check that the deformation of the component is within the limits of the design criteria.

Results and Discussion

The analysis shows that the von Mises equivalent stress has a maximum value of about190 MPa, which, compared with the material's yield strength of 350 MPa, results in a utilization factor of 54%.

The analysis also gives the maximum static displacements as 1.14 mm

Three different representations of the stress state are shown in Figure 3 through Figure 5.

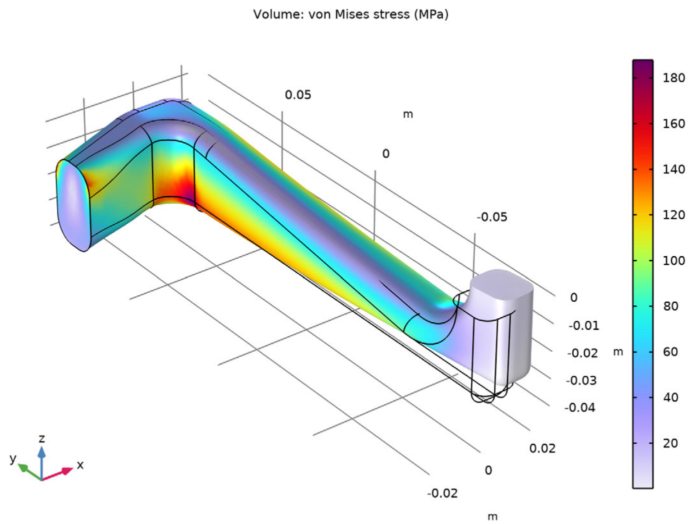


Figure 3: Equivalent stresses on the boundary of the domain.

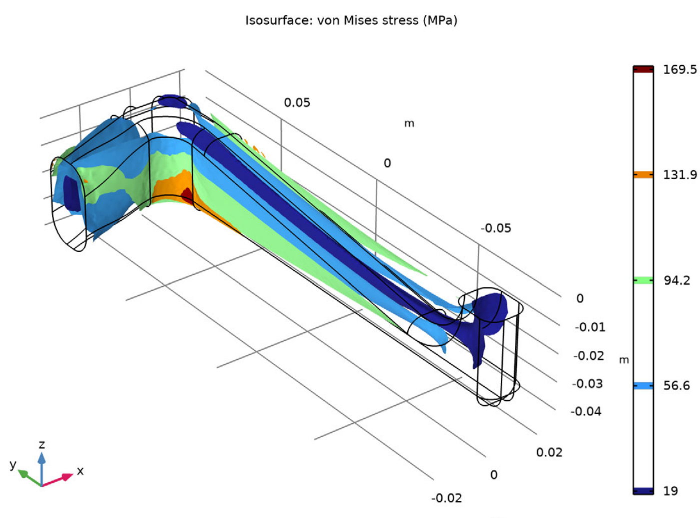


Figure 4: Isosurface plot of the equivalent stress.

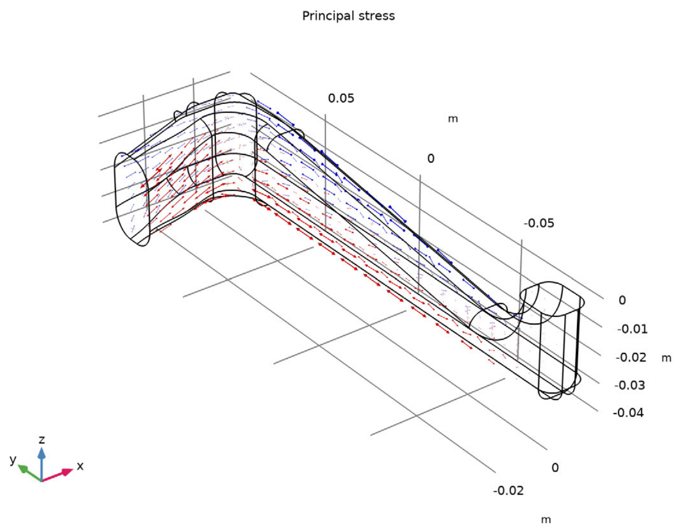


Figure 5: Arrow plot of the principal stresses.

Modeling Instructions

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 1 only.

Boundary Load 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2. Select Boundary 21 only.
3. In the **Settings** window for **Boundary Load**, locate the **Force** section.
4. Specify the $\mathbf{F}_A$ vector as

3[MPa]	x
0	y
3[MPa]	z

STUDY 1 (STATIC)

In this model, where there are many different studies, it is a good idea to assign manual names to some nodes in the model tree.

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

2 In the **Settings** window for **Study**, type `Study 1 (Static)` in the **Label** text field.

   The default settings in the generated solver are OK for this model, so it can be run directly.

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

Before moving on to analyzing the solution, rename the solver.

Solution, Static

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

2 In the **Settings** window for **Solution**, type `Solution, Static` in the **Label** text field.

RESULTS

Similarly, rename the solution dataset.

Static Solution

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1 (Static)/Solution, Static (sol1)**.

2 In the **Settings** window for **Solution**, type `Static Solution` in the **Label** text field.

   In the Results branch, you can create various plot types, evaluate expressions, or animate the results. The result features can visualize any expression containing, for example, the solution variables, their derivatives, and the space coordinates. Many frequently used expressions are predefined as postprocessing variables, and they are directly available in the **Expression** section menus for the various plot types.

   When the solver finishes, a default plot appears. It shows a volume plot of the von Mises stress with the deformed shape of the component. For future reference, you can rename it.

Static Stress Contour

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

2 In the **Settings** window for **3D Plot Group**, type `Static Stress Contour` in the **Label** text field.

Volume 1

1. In the **Model Builder** window, expand the **Static Stress Contour** node, then click **Volume 1**.
2. In the **Settings** window for **Volume**, locate the **Expression** section.
3. From the **Unit** list, choose **MPa**.
4. In the **Static Stress Contour** toolbar, click **Plot**.
5. Click the **Zoom Extents** button in the **Graphics** toolbar.

The applied loads are also available in a default plot.

Applied Loads, Static Solution

1. In the **Model Builder** window, under **Results** click **Applied Loads (solid)**.
2. In the **Settings** window for **Group**, type `Applied Loads, Static Solution` in the **Label** text field.

Boundary Loads (solid)

In the **Model Builder** window, expand the **Applied Loads, Static Solution** node.

In order not to hide any load vectors, a wireframe representation is used for the geometry as a default. The plot group is however prepared for a visualization with hidden surfaces.

Gray Surfaces

In the **Model Builder** window, expand the **Boundary Loads (solid)** node.

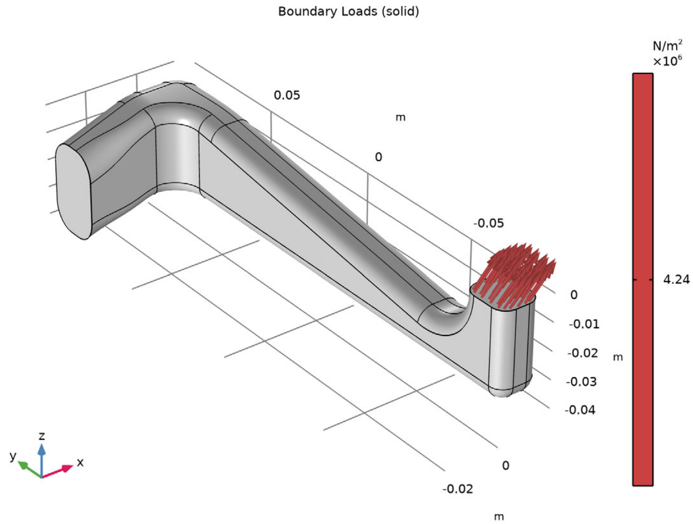
Transparency 1

1. In the **Model Builder** window, expand the **Gray Surfaces** node.
2. Right-click **Transparency 1** and choose **Disable**.

Boundary Loads (solid)

1. In the **Model Builder** window, under **Results>Applied Loads, Static Solution** click **Boundary Loads (solid)**.

2 In the **Boundary Loads (solid)** toolbar, click **Plot**.



To evaluate the maximum displacement, use a nonlocal maximum coupling.

DEFINITIONS

Maximum 1 (maxop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.
2 In the **Settings** window for **Maximum**, locate the **Source Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **All boundaries**.

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
U_max	maxop1(solid.disp)	m	Maximum deflection

Study 1 (Static)

Solution, Static (sol1)

1 In the **Model Builder** window, under **Study 1 (Static)>Solver Configurations** right-click **Solution, Static (sol1)** and choose **Solution>Update**.

This step is necessary in order to access variables that were created after the solution was performed.

Results

Global Evaluation 1

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

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Definitions>Variables>U_max - Maximum deflection - m**.

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

Expression	Unit	Description
`U_max`	`mm`	`Maximum deflection`

4 Click **Evaluate**.

The result, approximately 1.1 mm appears in the **Table** window.

Next, add a second plot group and create an isosurface plot. The resulting plot should resemble that in Figure 4.

Static Stress Isosurface

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Static Stress Isosurface` in the **Label** text field.

Isosurface 1

1 Right-click **Static Stress Isosurface** and choose **Isosurface**.

2 In the **Settings** window for **Isosurface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Stress>solid.mises - von Mises stress - N/m²**.

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

Deformation 1

1 Right-click **Isosurface 1** and choose **Deformation**.

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

With the following steps you can reproduce the principal stress arrow plot shown in Figure 5:

Static Principal Stress Arrow Plot

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Static Principal Stress Arrow Plot` in the **Label** text field.

Principal Stress Volume 1

1 In the **Static Principal Stress Arrow Plot** toolbar, click **More Plots** and choose **Principal Stress Volume**.

2 In the **Settings** window for **Principal Stress Volume**, locate the **Positioning** section.

3 Find the **X grid points** subsection. In the **Points** text field, type `10`.

4 Find the **Y grid points** subsection. In the **Points** text field, type `15`.

5 Find the **Z grid points** subsection. In the **Points** text field, type `10`.

In the **Static Principal Stress Arrow Plot** toolbar, click **Plot**.

Eigenfrequency Analysis

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

$$f = -\frac{-\lambda}{2\pi i}$$

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

If no damping is included in the material, the undamped natural frequencies are computed.

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

Results and Discussion

The first six eigenfrequencies are:

EIGENFREQUENCY	FREQUENCY
f_1	416 Hz
f_2	573 Hz
f_3	1924 Hz
f_4	2459 Hz
f_5	3112 Hz
f_6	3956 Hz

The mode shapes corresponding to the two lowest eigenfrequencies are shown in Figure 6. The deformed plot indicates an oscillation in the *xy*-plane for the lowest eigenfrequency, while the second lowest eigenmode shows an oscillation in the *yz*-plane.

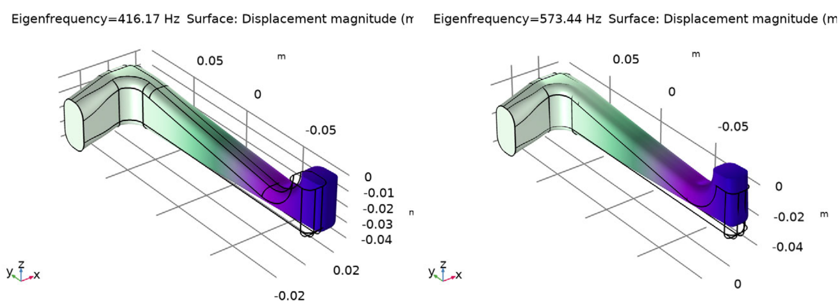


Figure 6: Eigenmodes of the two lowest eigenfrequencies.

Notes About the COMSOL Implementation

Any loads present on the model, such as the load from the static load case above, are ignored in the default eigenfrequency analysis. It is also possible to include effects from prestress. You can find an example of such an analysis in the example Vibrating String.

Modeling Instructions

Add a new study to your model.

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 (EIGENFREQUENCY)

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

2 In the **Settings** window for **Study**, type `Study 2 (Eigenfrequency)` in the **Label** text field.

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

Solution, Eigenfrequency

1 In the **Model Builder** window, expand the **Study 2 (Eigenfrequency)>Solver Configurations** node, then click **Solution 2 (sol2)**.

2 In the **Settings** window for **Solution**, type `Solution, Eigenfrequency` in the **Label** text field.

RESULTS

Eigenfrequency Solution

1 In the **Model Builder** window, under **Results>Datasets** click **Study 2 (Eigenfrequency)/Solution, Eigenfrequency (sol2)**.

2 In the **Settings** window for **Solution**, type `Eigenfrequency Solution` in the **Label** text field.

Undamped Mode Shapes

As a default, the first eigenmode is shown. Follow these steps to reproduce the plot in the left panel of Figure 6.

Take a look at the second mode as well.

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

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

3 From the **Eigenfrequency (Hz)** list, choose **573.44**.

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

Compare the resulting plot to that to the right in Figure 6.

You can give the plot a more descriptive name:

5 In the **Label** text field, type `Undamped Mode Shapes`.

Animation 1

In the **Undamped Mode Shapes** toolbar, click **Animation** and choose **Player**.

Mode Shape Animation

This creates an animation showing how the elbow bracket would deform if subjected to a harmonic load with a frequency near the selected eigenfrequency, in this case 571 Hz. To play the movie again, click the **Play** button in the **Graphics** toolbar.

The default animation sequence type when you add a player this way is **Dynamic data extension**. If you set the **Sequence type** to **Stored solutions** and then click the **Generate Frame** button, you get an animation where each frame corresponds to an eigenmode in the **Eigenfrequency** list. By using the **Frame number** slider in the **Frames** section you can then easily browse the eigenmodes.

Rename the player:

1 In the **Model Builder** window, expand the **Results>Undamped Mode Shapes** node, then click **Results>Export>Animation 1**.

2 In the **Settings** window for **Animation**, type `Mode Shape Animation` in the **Label** text field.

Damped Eigenfrequency Analysis

If the material has damping, the eigenvalue solver automatically switches to computation of the damped eigenfrequencies. The damped eigenfrequencies and eigenmodes are complex. The real part of the eigenfrequency corresponds to the frequency and the imaginary part represents the damping.

Results and Discussion

The first six eigenfrequencies are given below, and can be compared with the results from the undamped model.:

EIGENFREQUENCY	FREQUENCY	UNDAMPED FREQUENCY
f_1	414+41.3i Hz	416 Hz
f_2	571+56.9i Hz	573 Hz
f_3	1884+394i Hz	1924 Hz
f_4	2377+628i Hz	2459 Hz

EIGENFREQUENCY	FREQUENCY	UNDAMPED FREQUENCY
f_5	2950+992i Hz	3112 Hz
f_6	3623+1589i Hz	3956 Hz

The damping ratio of a certain mode is the ratio between the imaginary and the real part. It can be seen that the damping ratio increases rapidly as the natural frequency increases. This is an effect of the Rayleigh damping model.

You can find a table of the eigenfrequencies and corresponding damping in the evaluation group **Eigenfrequencies (Study 3 (Damped Eigenfrequency))**.

Modeling Instructions

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.

SOLID MECHANICS (SOLID)

Add damping.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** 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 From the **Input parameters** list, choose **Damping ratios**.

4 In the f_1 text field, type `400`.

5 In the ζ_1 text field, type `0.1`.

6 In the f_2 text field, type `600`.

7 In the ζ_2 text field, type `0.1`.

STUDY 3 (DAMPED EIGENFREQUENCY)

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

2 In the **Settings** window for **Study**, type `Study 3 (Damped Eigenfrequency)` in the **Label** text field.

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

Solution, Damped Eigenfrequency

1 In the **Model Builder** window, expand the **Study 3 (Damped Eigenfrequency)>Solver Configurations** node, then click **Solution 3 (sol3)**.

2 In the **Settings** window for **Solution**, type `Solution, Damped Eigenfrequency` in the **Label** text field.

RESULTS

Damped Eigenfrequency Solution

1 In the **Model Builder** window, under **Results>Datasets** click **Study 3 (Damped Eigenfrequency)/Solution, Damped Eigenfrequency (sol3)**.

2 In the **Settings** window for **Solution**, type `Damped Eigenfrequency Solution` in the **Label** text field.

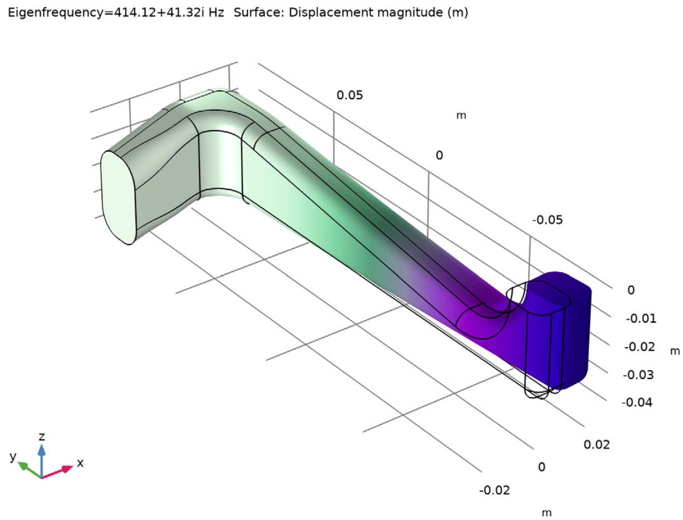
Damped Mode Shapes

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

2 In the **Settings** window for **3D Plot Group**, type `Damped Mode Shapes` in the **Label** text field.

The mode shape identical to the one obtained when solving the undamped problem. Only the frequency has changed.

Surface 1



The second study should still produce undamped eigenfrequencies when it is run next time, so you must make sure that the newly added Damping node is ignored.

STUDY 2 (EIGENFREQUENCY)

Step 1: Eigenfrequency

1. In the **Model Builder** window, expand the **Damped Mode Shapes** node, then click **Study 2 (Eigenfrequency)>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 tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Linear Elastic Material 1>Damping 1**.
5. Click **Disable**.

Time-Dependent Analysis

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

The purpose of this analysis is to find the transient response from a harmonic load during the first five periods. The excitation frequency is 500 Hz, which is between the first and second eigenfrequencies found in the eigenfrequency analysis.

This load is applied on the face indicated in Figure 2. The expression for the load can be written as

$$F_x = 1.5 \cdot [1 + \sin(2\pi \cdot 500\,\mathrm{Hz} \cdot t - \pi/2)]\ \ \mathrm{MPa}$$

where t denotes the time in seconds.

Results and Discussion

Because the loading is harmonic, the expected solution consists of an initial transient, and after long time the response is a stationary harmonic solution with its amplitude controlled by the damping of the system.

The following plot shows the x-displacement at a point on the loaded face:

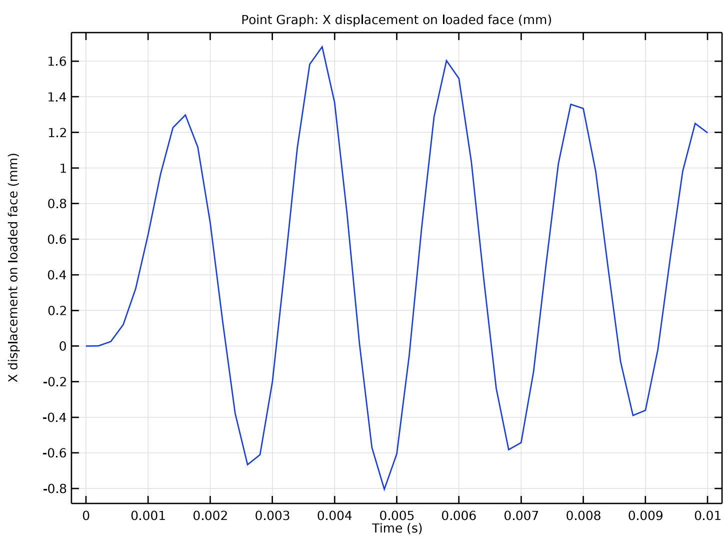


Figure 7: x-displacement at a point on the loaded face.

The figure below shows the von Mises stress in the bracket at 0.0036 s. The maximum value at this time is about 240 MPa.

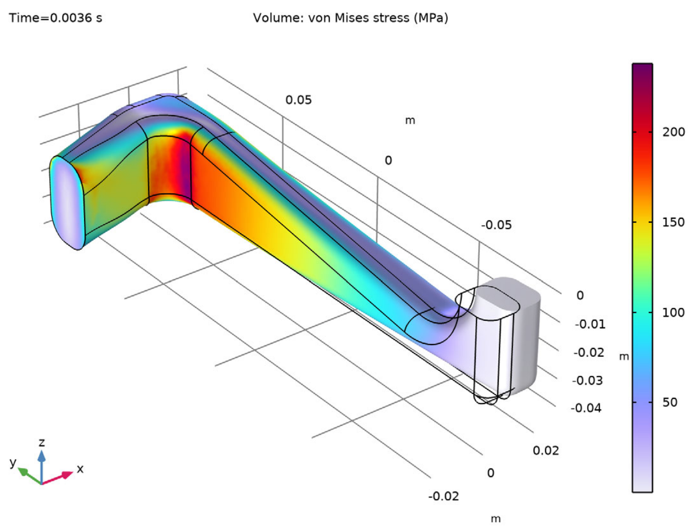


Figure 8: von Mises stress at t = 3.6 ms.

Notes About the COMSOL Implementation

When a harmonic load is used, the time step can sometimes oscillate in an inefficient manner, causing longer solution times. This can be avoided by using the more restrictive time stepping obtained by selecting the check box for **Time step increase delay**.

For more information on the settings for the time-dependent solver, see the *COMSOL Multiphysics Reference Manual*.

Modeling Instructions

If you are working from the beginning of this example, ignore the next two instructions. If you are starting from the short model version described in Static and Eigenfrequency Analyses of an Elbow Bracket, load that model as described here:

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **Structural Mechanics Module>Tutorials>elbow_bracket_brief** in the tree.

3 Click **Open**.

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> Time Dependent**.

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

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

STUDY 4

Solving for five periods with an excitation frequency of 500 Hz means solving for 10 ms. Save the solution every 0.2 ms.

Step 1: Time Dependent

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

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

3 In the **Output times** text field, type `range(0,2e-4,10e-3)`.

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

5 In the **Settings** window for **Study**, type `Study 4 (Time-Dependent)` in the **Label** text field.

Solution, Time-Dependent

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

2 In the **Settings** window for **Solution**, type `Solution, Time-Dependent` in the **Label** text field.

3 In the **Model Builder** window, expand the **Study 4 (Time-Dependent)> Solver Configurations>Solution, Time-Dependent (sol4)** node, then click **Time-Dependent Solver 1**.

4 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Absolute Tolerance** section.

5 From the **Tolerance method** list, choose **Manual**.

6 In the **Absolute tolerance** text field, type `1e-5`.

7 Click to expand the **Time Stepping** section. Select the **Time-step increase delay** check box. Keep the default value of 15.

This setting instructs the solver not to increase the time step until 15 consecutive steps have been successful.

8 In the **Amplification for high frequency** text field, type `0.95`.

By raising this value from its default value of 0.75 you reduce the damping of high frequencies.

You can reduce the file size significantly by not storing time derivatives when not needed.

9 Click to expand the **Output** section. Clear the **Store time derivatives** check box.

RESULTS

Before computing the solution, prepare a plot for displaying the results during the solution process.

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

Time-Dependent Solution

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 4 (Time-Dependent)/Solution, Time-Dependent (sol4)**.

2 In the **Settings** window for **Solution**, type `Time-Dependent Solution` in the **Label** text field.

Time-Dependent Displacement Graphs

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

2 In the **Settings** window for **1D Plot Group**, type `Time-Dependent Displacement Graphs` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Time-Dependent Solution (sol4)**.

Point Graph 1

1 Right-click **Time-Dependent Displacement Graphs** and choose **Point Graph**.

2 Select Point 30 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 (comp1)>Solid Mechanics>Displacement>Displacement field - m>u - Displacement field, X component**.

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

5 Select the **Description** check box.

6 In the associated text field, type `X displacement on loaded face`.

STUDY 4 (TIME-DEPENDENT)

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 4 (Time-Dependent)** 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.

4 From the **Plot group** list, choose **Time-Dependent Displacement Graphs**.

SOLID MECHANICS (SOLID)

A new load is needed for this study. You could just change the existing one, but when you have multiple studies it is better to have individual load nodes in the model tree and disable the ones not currently used in the study.

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

Static Load

1 In the **Model Builder** window, expand the **Solid Mechanics (solid)** node, then click **Boundary Load 1**.

2 In the **Settings** window for **Boundary Load**, type `Static Load` in the **Label** text field.

Time-Dependent Load

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

2 In the **Settings** window for **Boundary Load**, type `Time-Dependent Load` in the **Label** text field.

3 Select Boundary 21 only.

4 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

`1.5[MPa]*(1+sin(2*pi*500[Hz]*t-pi/2))`	x
0	y
0	z

STUDY 4 (TIME-DEPENDENT)

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 4 (Time-Dependent)** click **Step 1: Time Dependent**.

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

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

4 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Static Load**.

5 Click **Disable**.

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

RESULTS

Time-Dependent Displacement Graphs

Compare the result for the x-displacement with the graph shown in Figure 7.

Since a plot was added manually, default plots are not created for this study. Copy, and modify, the existing stress plot.

Time-Dependent Stress Contour

1 In the **Model Builder** window, right-click **Static Stress Contour** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Time-Dependent Stress Contour` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Time-Dependent Solution (sol4)**.

4 From the **Time (s)** list, choose **0.0036**.

5 In the **Time-Dependent Stress Contour** toolbar, click **Plot**.

Generate an animation of the solution.

Time-Dependent Stress Contour

1 In the **Time-Dependent Stress Contour** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, type `Time-Dependent Stress Contour` in the **Label** text field.

Time-Dependent Modal Analysis

In a modal-based analysis, the problem is reduced by representing the dynamics of the structure by a combination of a small number of its most significant eigenmodes. This is very efficient when the frequency content of the loads is limited, so that only a small number of modes are excited.

Results and Discussion

The plot in Figure 9 below shows the same x-displacement as in the previous section but with results from both the full and the modal based time-dependent analysis. The correspondence between the solutions is good even though only the first six eigenmodes are used.

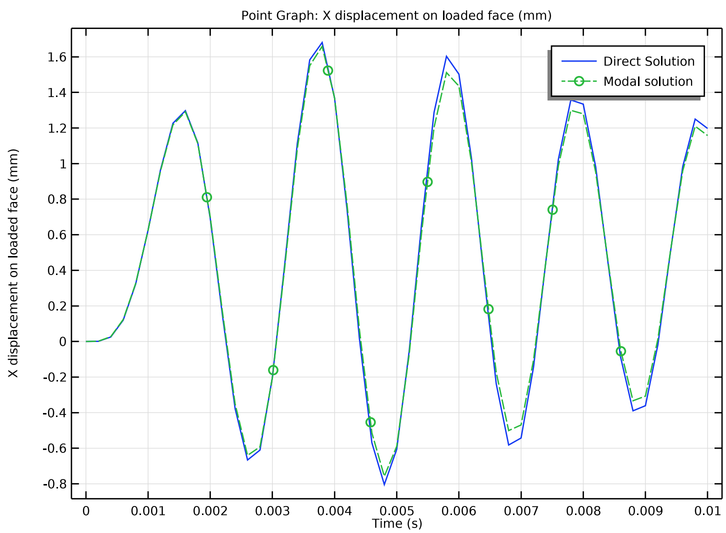


Figure 9: x-displacement at a point on the loaded surface for full and modal analyses.

Figure 10 below shows the von Mises stress in the bracket at 0.0036 s. The maximum value is 233 MPa, which can be compared with the 238 MPa computed using the direct solution above. In general, more modes than what is needed to compute accurate displacements are required to obtain good stress solutions.

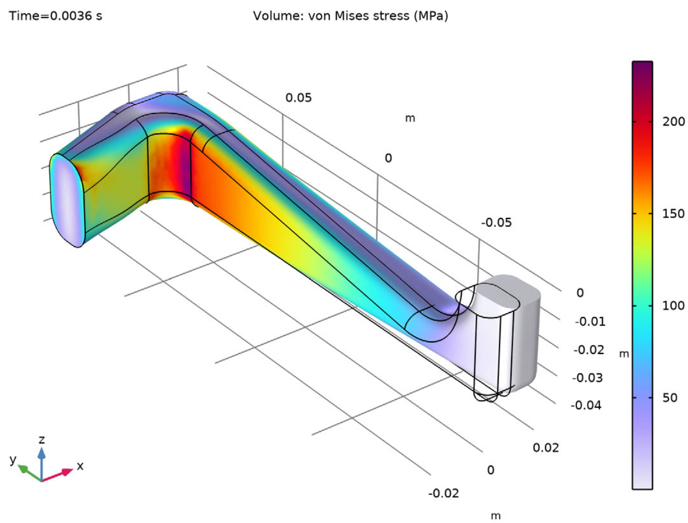


Figure 10: von Mises stress for modal solution.

Notes About the COMSOL Implementation

When you create a new study, it is possible to directly select **Time Dependent, Modal.** Such a study, however, generates a complete solver sequence including the eigenvalue computation step. Because the eigenvalues are already available, you create an **Empty Study** and add the study steps manually.

The undamped eigenmodes are used as the base, and the damping is provided by the material.

In the modal time-dependent procedure, all loads must have the same variation in time, specified in the study step. This means that you should not enter any time-dependent loads (that is, loads with an explicit dependency on the time variable `t`). If you are in a situation where all loads do not have the same temporal variation, you can instead use a more general approach available in the reduced order modeling framework.

Modeling Instructions

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 **Empty Study**.

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

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

SOLID MECHANICS (SOLID)

Modal Time-Dependent Load

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

2 In the **Settings** window for **Boundary Load**, type `Modal Time-Dependent Load` in the **Label** text field.

3 Select Boundary 21 only.

4 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

`1.5[MPa]`	x
`0`	y
`0`	z

STUDY 5 (MODAL TIME-DEPENDENT)

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

2 In the **Settings** window for **Study**, type `Study 5 (Modal Time-Dependent)` in the **Label** text field.

Time Dependent, Modal

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent, Modal**.

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

3 In the **Output times** text field, type `range(0,2e-4,10e-3)`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Static Load** and **Component 1 (Comp1)>Solid Mechanics (Solid)>Time-Dependent Load**.

6 Click **Disable**.

Solution, Modal Time-Dependent

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

2 In the **Settings** window for **Solution**, type `Solution, Modal Time-Dependent` in the **Label** text field.

3 In the **Model Builder** window, expand the **Study 5 (Modal Time-Dependent)>Solver Configurations>Solution, Modal Time-Dependent (sol5)** node, then click **Modal Solver 1**.

4 In the **Settings** window for **Modal Solver**, click to expand the **Absolute Tolerance** section.

5 From the **Tolerance method** list, choose **Manual**.

6 In the **Absolute tolerance** text field, type `1e-5`.

7 Locate the **Eigenpairs** section. From the **Solution** list, choose **Solution, Eigenfrequency (sol2)**.

8 Click to expand the **Advanced** section. In the **Load factor** text field, type `1+sin(2*pi*500[Hz]*t-pi/2)`.

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

RESULTS

Modal Time-Dependent Solution

1 In the **Model Builder** window, under **Results>Datasets** click **Study 5 (Modal Time-Dependent)/Solution, Modal Time-Dependent (sol5)**.

2 In the **Settings** window for **Solution**, type `Modal Time-Dependent Solution` in the **Label** text field.

Reproduce the plot in Figure 10 by following these instructions:

Modal Time-Dependent Stress Contour

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

2 In the **Settings** window for **3D Plot Group**, type `Modal Time-Dependent Stress Contour` in the **Label** text field.

3 Locate the **Data** section. From the **Time (s)** list, choose **0.0036**.

Volume 1

1 In the **Model Builder** window, expand the **Modal Time-Dependent Stress Contour** node, then click **Volume 1**.

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

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

4 In the **Modal Time-Dependent Stress Contour** toolbar, click **Plot**.

Add the results from this study to the graph of the direct time dependent results, so that the methods can be compared.

Point Graph 1

1 In the **Model Builder** window, under **Results>Time-Dependent Displacement Graphs** click **Point Graph 1**.

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

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

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

5 In the table, enter the following settings:

Legends
Direct Solution

Point Graph 2

1 Right-click **Results>Time-Dependent Displacement Graphs>Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Modal Time-Dependent Solution (sol5)**.

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

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

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

Legends
Modal solution

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

In the **Time-Dependent Displacement Graphs** toolbar, click **Plot**.

Frequency Response Analysis

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

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

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

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

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

You can visualize the amplitudes and phases as well as the solution at a specific angle (time). When plotting, COMSOL Multiphysics multiplies the solution by $e^{i\phi}$, where ϕ is the angle specified in the **Solution at angle (phase)** text field in the settings for the dataset. The plot then shows the real part of the evaluated expression

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

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

The purpose of this analysis is to find the response from a harmonic load with an excitation frequency in the range 350–650 Hz, which includes the first two eigenfrequencies found in the eigenfrequency analysis. The load amplitudes are

$$F_x = 3 \cdot 10^6 \ \mathrm{N/m^2}$$

$$F_z = 3 \cdot 10^6 \ \mathrm{N/m^2}$$

and there is no phase shift between the load components.

Results and Discussion

The amplitudes of the x-, y-, and z-displacements as functions of excitation frequency, at a point on the face where the load is applied, appear in the following figure:

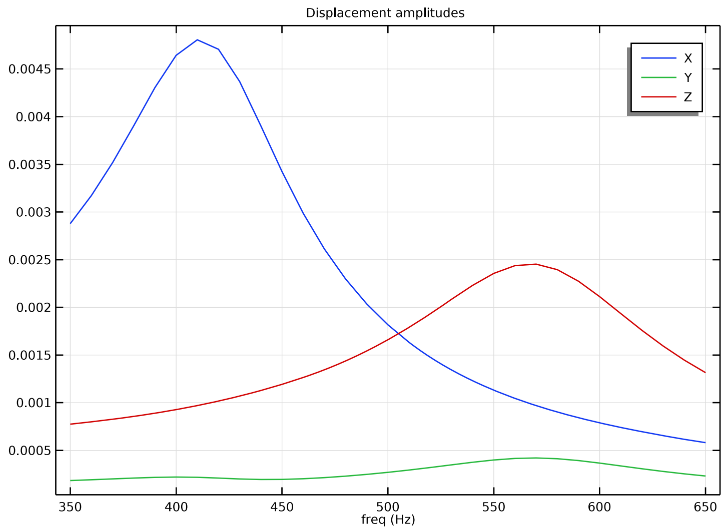


Figure 11: Displacement amplitudes vs. excitation frequency.

The peaks in the displacement amplitude curves are associated with the two lowest eigenfrequencies of the bracket. Note that the lowest eigenfrequency, 416 Hz, corresponds to the peak on the x-displacement amplitude curve, while the next eigenfrequency, 573 Hz, corresponds to the peak on the z-displacement amplitude curve. This can be expected based on the eigenmode shapes obtained in the eigenfrequency analysis.

In Figure 12, the default stress plot is shown. For a frequency domain analysis, it shows the maximum von Mises stress in each point during the cycle.

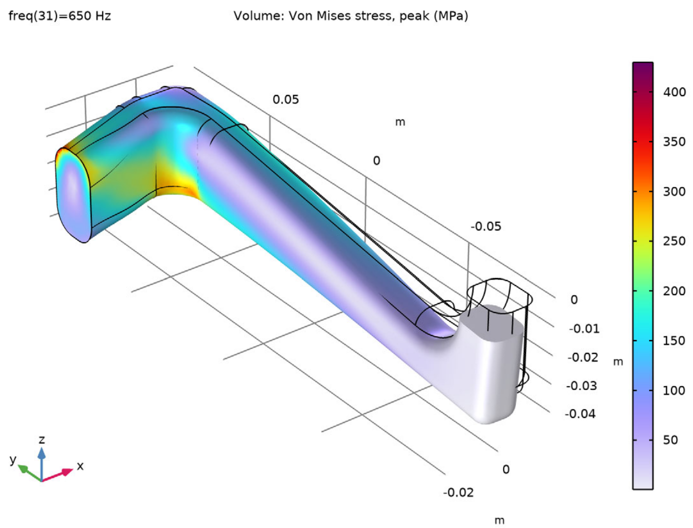


Figure 12: Maximum equivalent stress during the cycle at the last frequency, 650 Hz-

Notes About the COMSOL Implementation

The loads are given without explicit time dependencies, since the harmonic variation is an underlying assumption in this analysis type. The loads can have different phase angles. This can either be obtained by adding a **Phase** subnode under the load, or by writing the load on a complex form.

Usually, when performing a frequency response analysis, you want to sweep over a frequency range. This can be done using the parametric solver, using the frequency as a parameter.

Modeling Instructions

Applied Loads (solid)

In the **Model Builder** window, under **Results** right-click **Applied Loads (solid)** and choose **Delete**.

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>Frequency Domain**.

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

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

STUDY 6 (FREQUENCY DOMAIN)

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

2 In the **Settings** window for **Study**, type `Study 6 (Frequency Domain)` in the **Label** text field.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 6 (Frequency Domain)** click **Step 1: Frequency Domain**.

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

3 In the **Frequencies** text field, type `range(350,10,650)`.

The load is the same as in the stationary study, so you can reuse the first load in the model tree.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Time-Dependent Load** and **Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Time-Dependent Load**.

6 Click **Disable**.

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

Solution, Frequency Domain

1 In the **Model Builder** window, expand the **Study 6 (Frequency Domain)>Solver Configurations** node, then click **Solution 6 (sol6)**.

2 In the **Settings** window for **Solution**, type `Solution, Frequency Domain` in the **Label** text field.

RESULTS

Frequency Domain Solution

1 In the **Model Builder** window, under **Results>Datasets** click **Study 6 (Frequency Domain)/Solution, Frequency Domain (sol6)**.

2 In the **Settings** window for **Solution**, type `Frequency Domain Solution` in the **Label** text field.

Frequency-Response Stress Contour

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

2 In the **Settings** window for **3D Plot Group**, type `Frequency-Response Stress Contour` in the **Label** text field.

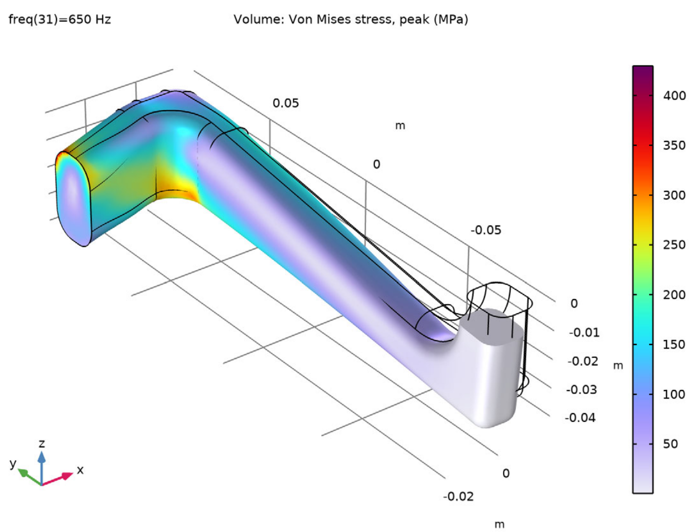
Volume 1

1 In the **Model Builder** window, expand the **Frequency-Response Stress Contour** node, then click **Volume 1**.

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

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

4 In the **Frequency-Response Stress Contour** toolbar, click **Plot**.



Add a 1D plot group and reproduce the displacement amplitude graphs in Figure 11.

Applied Loads (solid)

In the **Model Builder** window, under **Results** right-click **Applied Loads (solid)** and choose **Delete**.

Frequency Response Displacement Graphs

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 Response Displacement Graphs` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Frequency Domain Solution (sol6)**.

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

5 In the **Title** text area, type `Displacement amplitudes`.

Point Graph 1

1 Right-click **Frequency Response Displacement Graphs** and choose **Point Graph**.

2 Select Point 29 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 (comp1)>Solid Mechanics>Displacement>Displacement amplitude (material and geometry frames) - m>solid.uAmpX - Displacement amplitude, X component**.

4 Locate 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
X

Point Graph 2

1 Right-click **Point Graph 1** 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 (comp1)>Solid Mechanics>Displacement>Displacement amplitude (material and geometry frames) - m>solid.uAmpY - Displacement amplitude, Y component**.

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

Legends
Y

Point Graph 3

1 Right-click **Point Graph 2** 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 (comp1)>Solid Mechanics>Displacement>Displacement amplitude (material and geometry frames) - m>solid.uAmpZ - Displacement amplitude, Z component**.

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

Legends
Z

4 In the **Frequency Response Displacement Graphs** toolbar, click **Plot**.

Compare the resulting plot with that in Figure 11.

Frequency Response Modal Analysis

You can also solve the same frequency response problem using the modal superposition method. The same remarks as for the time-dependent modal analysis above are relevant.

Results and Discussion

In Figure 13 below, the results from the modal frequency response analysis are overlaid on the results from the previous direct frequency response analysis (see Figure 11). The curves are almost indistinguishable. The response is to a large degree controlled by the lowest eigenmodes, which are used in the modal analysis. The modal method is however much more efficient in terms of computer resources.

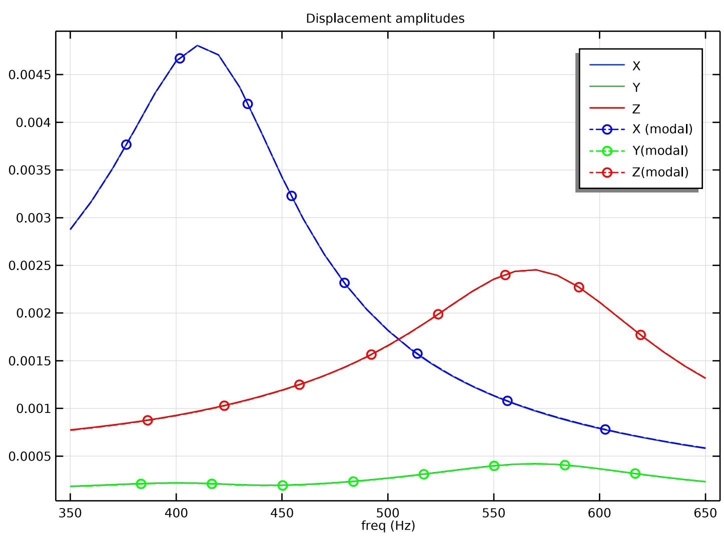


Figure 13: Displacement amplitudes vs. excitation frequency for direct and modal frequency-response analyses.

SOLID MECHANICS (SOLID)

Modal Frequency Load

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

2 In the **Settings** window for **Boundary Load**, type `Modal Frequency Load` in the **Label** text field.

3 Right-click **Modal Frequency Load** and choose **Harmonic Perturbation**.

The modal frequency response is a perturbation type of analysis. The load must then be declared as a perturbation load.

4 Select Boundary 21 only.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

3[MPa]	x
0	y
3[MPa]	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 **Empty Study**.

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

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

STUDY 7 (MODAL FREQUENCY RESPONSE)

In the **Settings** window for **Study**, type `Study 7 (Modal Frequency Response)` in the **Label** text field.

Frequency Domain, Modal

1 In the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Frequency Domain, Modal**.

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

3 In the **Frequencies** text field, type `range(350,10,650)`.

Since only loads having a harmonic perturbation are used in this study type, only the last added load, which is declared as a perturbation load, will be taken into account. For clarity, you can still disable all other loads.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Static Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Time-Dependent Load**, and **Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Time-Dependent Load**.

6 Click **Disable**.

Solution, Modal Frequency-Domain

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

2 In the **Settings** window for **Solution**, type `Solution, Modal Frequency-Domain` in the **Label** text field.

3 In the **Model Builder** window, expand the **Study 7 (Modal Frequency Response)> Solver Configurations>Solution, Modal Frequency-Domain (sol7)** node, then click **Modal Solver 1**.

4 In the **Settings** window for **Modal Solver**, locate the **Eigenpairs** section.

5 From the **Solution** list, choose **Solution, Eigenfrequency (sol2)**.

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

RESULTS

Modal Frequency-Domain Solution

1 In the **Model Builder** window, under **Results>Datasets** click **Study 7 (Modal Frequency Response)/Solution, Modal Frequency-Domain (sol7)**.

2 In the **Settings** window for **Solution**, type `Modal Frequency-Domain Solution` in the **Label** text field.

Modal Frequency-Response Stress Contour

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

2 In the **Settings** window for **3D Plot Group**, type `Modal Frequency-Response Stress Contour` in the **Label** text field.

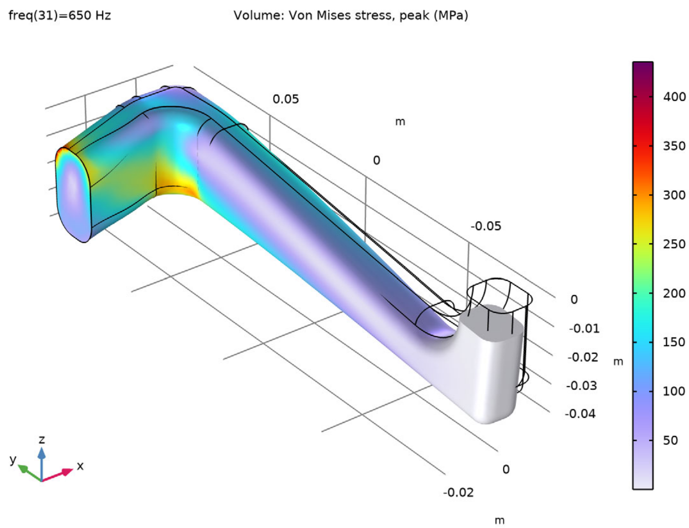
Volume 1

1 In the **Model Builder** window, expand the **Modal Frequency-Response Stress Contour** node, then click **Volume 1**.

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

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

4 In the **Modal Frequency-Response Stress Contour** toolbar, click **Plot**.



Reproduce the plot in Figure 13 as follows:

Point Graph 4

1 In the **Model Builder** window, under **Results>Frequency Response Displacement Graphs** 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 **Modal Frequency-Domain Solution (sol7)**.

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

5 Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

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

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

Legends
`X (modal)`

Point Graph 5

1 Right-click **Point Graph 4** 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 (comp1)> Solid Mechanics>Displacement>Displacement amplitude (material and geometry frames) - m>solid.uAmpY - Displacement amplitude, Y component**.

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

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

Legends
`Y(modal)`

Point Graph 6

1 Right-click **Point Graph 5** 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 (comp1)> Solid Mechanics>Displacement>Displacement amplitude (material and geometry frames) - m>solid.uAmpZ - Displacement amplitude, Z component**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

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

Legends
`Z(modal)`

5 In the **Frequency Response Displacement Graphs** toolbar, click **Plot**.

Parametric Analysis

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

The purpose of this example is to find the response to static loading of the bracket as a function of the direction of the load parameterized by the angle α.

Apply the load on the face shown in Figure 2. To control the direction of the load, introduce a parameter in the load expressions:

$$F_x = 3 \cdot \cos(\alpha \cdot \pi / 180) \text{ MPa}$$
$$F_z = 3 \cdot \sin(\alpha \cdot \pi / 180) \text{ MPa}$$

where α is the angle of the load direction in the xz-plane. Let $-45° \le \alpha \le 45°$.

Results and Discussion

The following plot shows the x-, y-, and z-displacements as functions of the direction of the load, α, at a point on the surface where the load is applied:

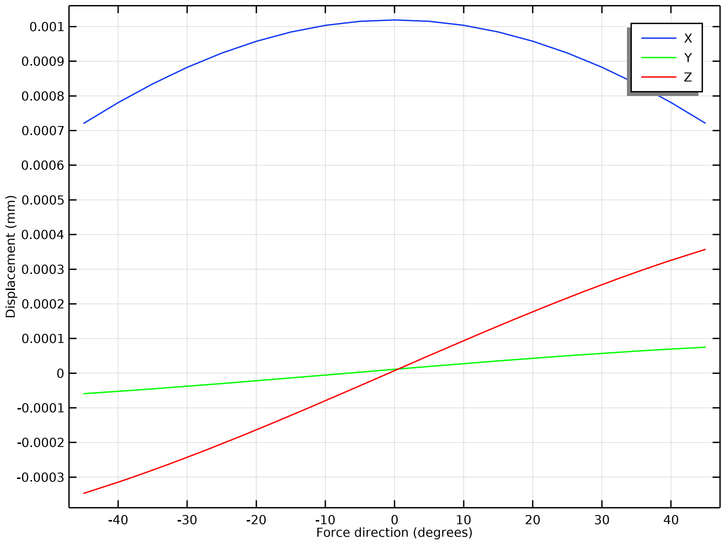


Figure 14: Displacement amplitudes versus load direction.

Notes About the COMSOL Implementation

When you perform a parametric study, you will select the underlying study type (here: Stationary) first. Then you add a Parametric feature, and define the parameter values.

Modeling Instructions

Applied Loads (solid)

In the **Model Builder** window, under **Results** right-click **Applied Loads (solid)** and choose **Delete**.

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 8 (PARAMETRIC STATIC)

1 In the **Model Builder** window, click **Study 8**.

2 In the **Settings** window for **Study**, type `Study 8 (Parametric Static)` in the **Label** text field.

Parametric Sweep

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

It is necessary to define the parameters to be used. The values given here do not influence the parametric solver, but can be used for studies outside it.

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**
`alpha`	`-45`	-45	`Solver parameter`

Add a load with a parameter dependency.

SOLID MECHANICS (SOLID)

Parametric Load

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

2 In the **Settings** window for **Boundary Load**, type `Parametric Load` in the **Label** text field.

3 Select Boundary 21 only.

4 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

`3[MPa]*cos(alpha*pi/180)`	x
`0`	y
`3[MPa]*sin(alpha*pi/180)`	z

STUDY 8 (PARAMETRIC STATIC)

Parametric Sweep

1 In the **Model Builder** window, under **Study 8 (Parametric Static)** 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**
alpha (Solver parameter)	`range(-45,5,45)`	

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 tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Static Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Time-Dependent Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Time-Dependent Load**, and **Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Frequency Load**.

5 Click **Disable**.

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

Solution, Parametric Static

1 In the **Model Builder** window, expand the **Study 8 (Parametric Static)> Solver Configurations** node, then click **Solution 8 (sol8)**.

2 In the **Settings** window for **Solution**, type `Solution, Parametric Static` in the **Label** text field.

RESULTS

Parametric Static Solution

1 In the **Model Builder** window, under **Results>Datasets** click **Study 8 (Parametric Static)/ Solution, Parametric Static (sol8)**.

2 In the **Settings** window for **Solution**, type `Parametric Static Solution` in the **Label** text field.

Stress (solid)

In the **Model Builder** window, under **Results** right-click **Stress (solid)** and choose **Delete**.

Applied Loads, Parametric Solution

1 In the **Model Builder** window, under **Results** click **Applied Loads (solid)**.
2 In the **Settings** window for **Group**, type `Applied Loads, Parametric Solution` in the **Label** text field.
3 In the **Model Builder** window, expand the **Applied Loads, Parametric Solution** node.

Boundary Loads (solid) 1

In the **Model Builder** window, expand the **Results>Applied Loads, Parametric Solution> Boundary Loads (solid) 1** node.

Transparency 1

1 In the **Model Builder** window, expand the **Results>Applied Loads, Parametric Solution> Boundary Loads (solid) 1>Gray Surfaces** node.
2 Right-click **Transparency 1** and choose **Disable**.

RESULTS

Applied Loads, Parametric Solution

1 In the **Model Builder** window, collapse the **Results>Applied Loads, Parametric Solution** node.
2 In the **Model Builder** window, click **Results**.
3 Click **Yes** to confirm.

Parametric Response Graphs

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Parametric Response Graphs` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Parametric Static Solution (sol8)**.
4 Locate the **Title** section. From the **Title type** list, choose **None**.
5 Locate the **Plot Settings** section. Select the **x-axis label** check box.
6 In the associated text field, type `Force direction (degrees)`.
7 Select the **y-axis label** check box.
8 In the associated text field, type `Displacement (mm)`.

Point Graph 1

1 Right-click **Parametric Response Graphs** and choose **Point Graph**.
2 Select Point 29 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 **Legends** section. Select the **Show legends** check box.

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

7 In the table, enter the following settings:

Legends
X

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.

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

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

Legends
Y

Point Graph 3

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

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

3 In the **Expression** text field, type w.

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

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

Legends
Z

In the **Parametric Response Graphs** toolbar, click **Plot**.

Linear Buckling Analysis

A structure under compression can sometimes become unstable due to buckling. The critical buckling load can be estimated using a linear buckling analysis.

To perform this analysis, you first run a stress analysis with an arbitrary load level. In a second study step, the buckling load is computed as a scale factor with respect to the load used in the first analysis.

Results and Discussion

The computed eigenvalue is 103. Since the load applied in the stationary step was 1 kN, the estimated buckling load is 103 kN. The shape of the buckling mode is shown below.

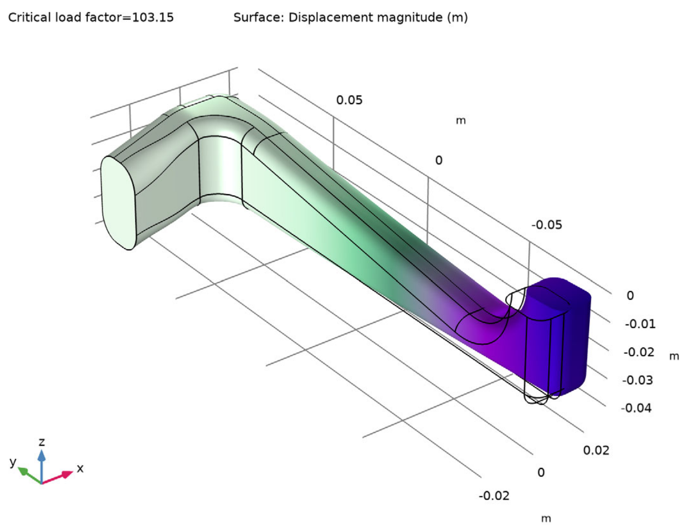


Figure 15: Buckling mode shape.

It can be noted that the stresses caused by the preload are so large in this structure that a plastic collapse would occur long before the buckling load was reached.

Notes About the COMSOL Implementation

When a Linear Buckling study is selected, both study steps are automatically prepared. It is only necessary to define the reference load.

Modeling Instructions

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.

SOLID MECHANICS (SOLID)

Next, set up the buckling preload.

Buckling Preload

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

2 In the **Settings** window for **Boundary Load**, type `Buckling Preload` in the **Label** text field.

3 Select Boundary 21 only.

4 Locate the **Force** section. From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{\mathrm{tot}}$ vector as

0	x
1[kN]	y
0	z

STUDY 9 (LINEAR BUCKLING)

1 In the **Model Builder** window, click **Study 9**.

2 In the **Settings** window for **Study**, type `Study 9 (Linear Buckling)` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 9 (Linear Buckling)** 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 tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Static Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Time-Dependent Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Time-Dependent Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Frequency Load**, and **Component 1 (Comp1)>Solid Mechanics (Solid)>Parametric Load**.

5 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 tree, select
Component 1 (Comp1)>Solid Mechanics (Solid)>Static Load,
Component 1 (Comp1)>Solid Mechanics (Solid)>Time-Dependent Load,
Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Time-Dependent Load,
Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Frequency Load, and
Component 1 (Comp1)>Solid Mechanics (Solid)>Parametric Load.

5 Click **Disable**.

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

Solution, Linear Buckling

1 In the **Model Builder** window, expand the **Study 9 (Linear Buckling)>Solver Configurations** node, then click **Solution 9 (sol9)**.

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

RESULTS

Linear Buckling Solution

1 In the **Model Builder** window, under **Results>Datasets** click **Study 9 (Linear Buckling)/ Solution, Linear Buckling (sol9)**.

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

Linear Buckling Preload Solution

1 In the **Model Builder** window, under **Results>Datasets** click **Study 9 (Linear Buckling)/ Solution Store 1 (sol10)**.

2 In the **Settings** window for **Solution**, type `Linear Buckling Preload Solution` in the **Label** text field.

With the following steps you can reproduce the plot in Figure 15:

Surface 1

1. In the **Model Builder** window, expand the **Mode Shape (solid)** node, then click **Surface 1**.
2. In the **Mode Shape (solid)** toolbar, click **Plot**.

Buckling Shape Plot

1. In the **Model Builder** window, under **Results** click **Mode Shape (solid)**.
2. In the **Settings** window for **3D Plot Group**, type `Buckling Shape Plot` in the **Label** text field.

Make all the studies possible to repeat by disabling loads which were added since each study was analyzed.

STUDY 1 (STATIC)

Step 1: Stationary

1. In the **Model Builder** window, expand the **Study 1 (Static)** node, then 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 tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Time-Dependent Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Time-Dependent Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Frequency Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Parametric Load**, and **Component 1 (Comp1)>Solid Mechanics (Solid)>Buckling Preload**.
5. Click **Disable**.

STUDY 4 (TIME-DEPENDENT)

Step 1: Time Dependent

1. In the **Model Builder** window, under **Study 4 (Time-Dependent)** click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.
3. In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Time-Dependent Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Frequency Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Parametric Load**, and **Component 1 (Comp1)>Solid Mechanics (Solid)>Buckling Preload**.

4 Click **Disable**.

STUDY 5 (MODAL TIME-DEPENDENT)

Step 1: Time Dependent, Modal

1 In the **Model Builder** window, under **Study 5 (Modal Time-Dependent)** click **Step 1: Time Dependent, Modal**.

2 In the **Settings** window for **Time Dependent, Modal**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Frequency Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Parametric Load**, and **Component 1 (Comp1)>Solid Mechanics (Solid)>Buckling Preload**.

4 Click **Disable**.

STUDY 6 (FREQUENCY DOMAIN)

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 6 (Frequency Domain)** click **Step 1: Frequency Domain**.

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

3 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Modal Frequency Load**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Parametric Load**, and **Component 1 (Comp1)>Solid Mechanics (Solid)>Buckling Preload**.

4 Click **Disable**.

STUDY 7 (MODAL FREQUENCY RESPONSE)

Step 1: Frequency Domain, Modal

1 In the **Model Builder** window, under **Study 7 (Modal Frequency Response)** click **Step 1: Frequency Domain, Modal**.

2 In the **Settings** window for **Frequency Domain, Modal**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Parametric Load** and **Component 1 (Comp1)>Solid Mechanics (Solid)>Buckling Preload**.

4 Click **Disable**.

STUDY 8 (PARAMETRIC STATIC)

Step 1: Stationary

1 In the **Model Builder** window, under **Study 8 (Parametric Static)** click **Step 1: Stationary**.

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

3 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Buckling Preload**.

4 Click **Disable**.