Created in COMSOL Multiphysics 6.1

Static and Eigenfrequency Analyses of an Elbow Bracket

Introduction

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

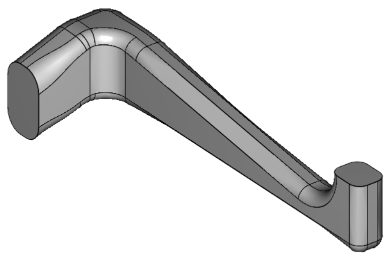


Figure 1: Geometry of the elbow bracket.

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

- Static analysis
- Eigenfrequency analysis
- Damped eigenfrequency analysis

In an extended version of this model, also the following study types are covered:

- 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.

Each of the listed analysis types corresponds to a *study* type; the available studies 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.

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.

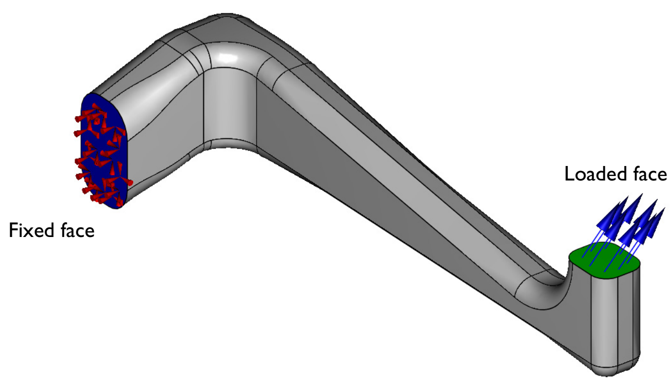


Figure 2: Constraint and loading of the bracket.

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_brief`

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**.

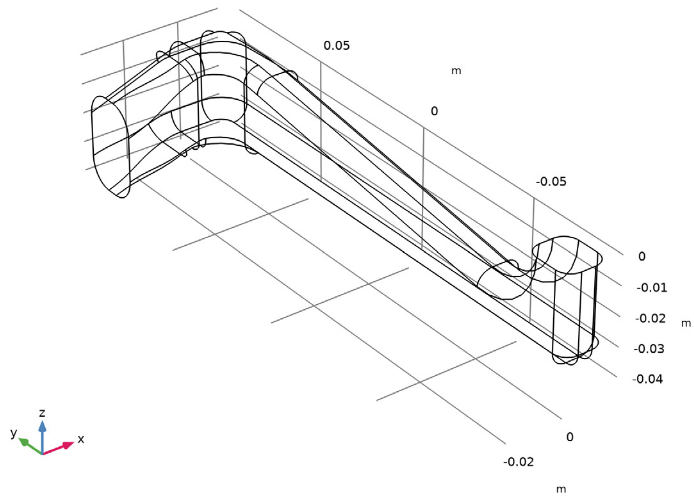
4 Click **Browse**.

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

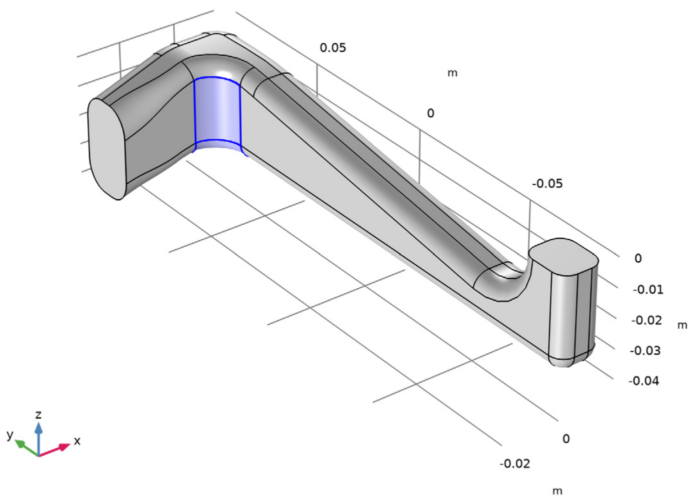
1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.

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.
5. 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 about 190 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.

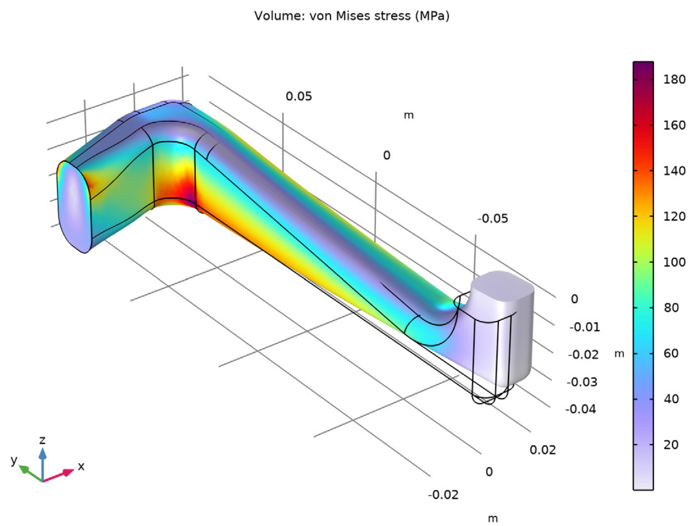


Figure 3: Equivalent stresses on the boundary of the domain.

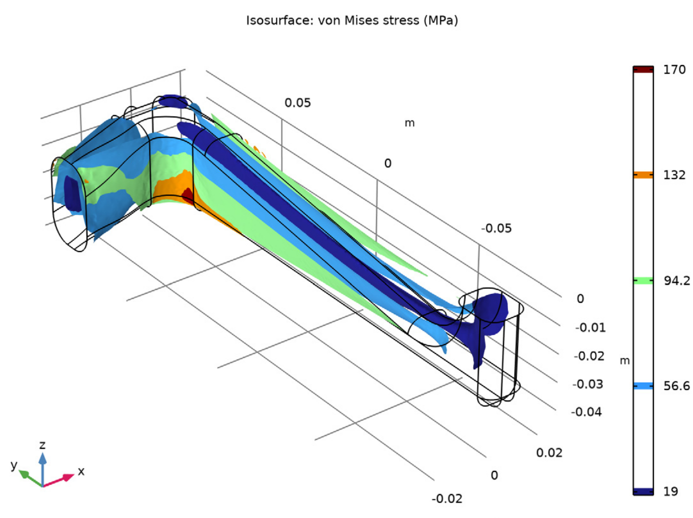


Figure 4: Isosurface plot of the equivalent stress.

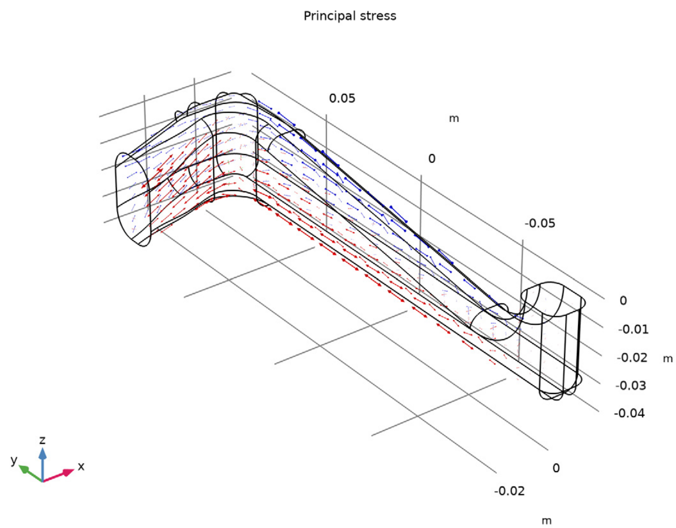


Figure 5: Arrow plot of the principal stresses.

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 available as a predefined plot.
6 In the **Home** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.
2 In the tree, select **Static Solution (sol1)>Solid Mechanics>Applied Loads (solid)>Boundary Loads (solid)**.
3 Click **Add Plot** in the window toolbar.
4 In the **Home** toolbar, click **Add Predefined Plot**.

RESULTS

Applied Loads, Static Solution

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

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 **Applied Loads, Static Solution** node.

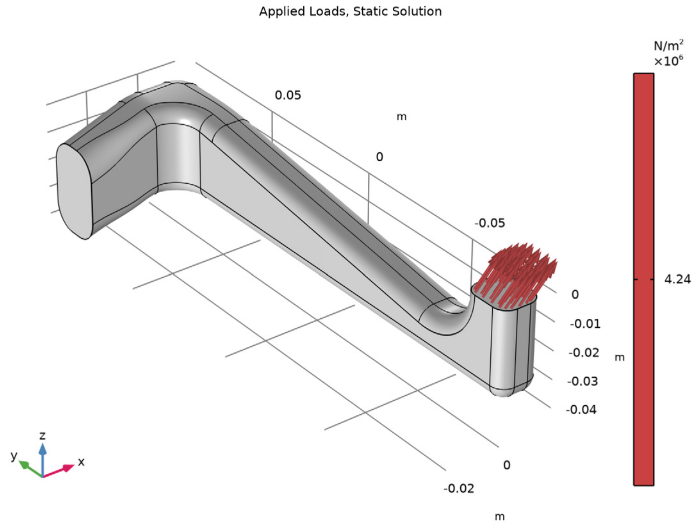
Transparency 1

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

Applied Loads, Static Solution

1 In the **Model Builder** window, under **Results** click **Applied Loads, Static Solution**.

2 In the **Applied Loads, Static Solution** 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 a= **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`.

6 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.

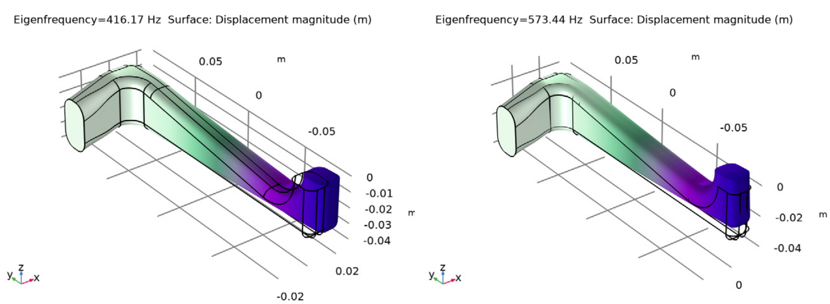


Figure 6: Eigenmodes of the two lowest eigenfrequencies.

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 (rounded to three digits) 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
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

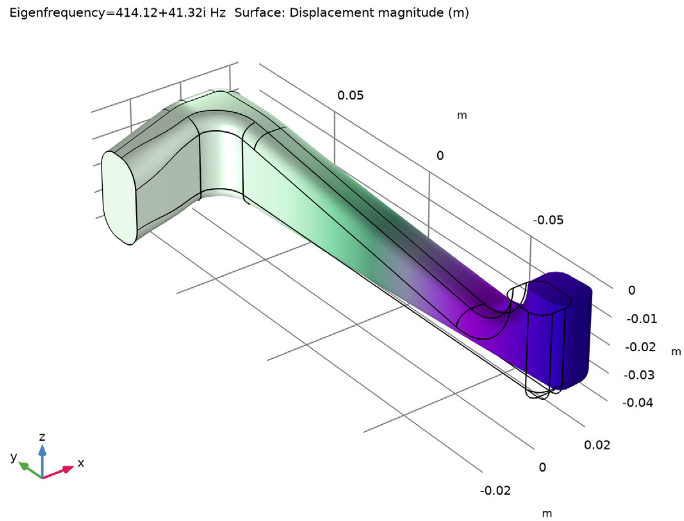
1 In the **Model Builder** window, under **Results>Datasets** click **Study 3 (Damped Eigenfrequency)/Solution, Damped Eigenfrequency (sol3)**.
2 In the **Settings** window for **Solution**, type `Damped Eigenfrequency Solution` in the **Label** text field.

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**.