Created in COMSOL Multiphysics 6.1

Vibrating Membrane

Introduction

In the following example, you compute the natural frequencies of a pretensioned membrane using the 3D Membrane interface. This is an example of “stress stiffening”; where the transverse stiffness of a membrane is directly proportional to the tensile force.

The results are compared with the analytical solution.

Model Definition

The model consists of a circular membrane, supported along its outer edge.

GEOMETRY

- Membrane radius, R = 0.25 m
- Membrane thickness, h = 0.2 mm

MATERIAL

- Young’s modulus, E = 200 GPa
- Poisson’s ratio, $\nu = 0.33$
- Mass density, $\rho = 7850\ \mathrm{kg/m^3}$

CONSTRAINTS

The outer edge of the membrane is supported in the transverse direction. Two points have constraints in the in-plane direction in order to avoid rigid body motions.

LOAD

The membrane is pretensioned by in the radial direction with σ_i = 100 MPa, giving a membrane force T_0 = 20 kN/m.

Results and Discussion

The analytical solution for the natural frequencies of the vibrating membrane given in Ref. 1 is:

$$f_{ij} = \frac{k_{ij}}{2\pi R}\sqrt{\frac{T_0}{h\rho}} \tag{1}$$

The values k_{ij} are derived from the roots of the Bessel functions of the first kind.

In Table 1 the computed results are compared with the results from Equation 1. The agreement is very good. The mode shapes for the first six modes are shown in Figure 1 through Figure 6. Note that some of the modes have duplicate eigenvalues, which is a common property for structures with symmetries.

TABLE 1: COMPARISON BETWEEN ANALYTICAL AND COMPUTED NATURAL FREQUENCIES.

Mode number	**Factor**	**Analytical frequency (Hz)**	**COMSOL result (Hz)**
1	k_{10} = 2.4048	172.8	172.8
2	k_{11} = 3.8317	275.3	275.3
3	k_{11} = 3.8317	275.3	275.3
4	k_{12} = 5.1356	369.0	369.0
5	k_{12} = 5.1356	369.0	369.0
6	k_{20} = 5.5201	396.6	396.7

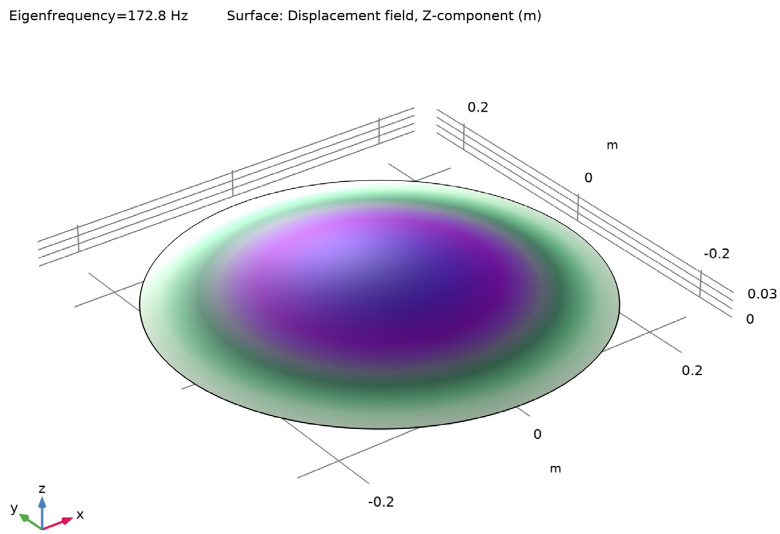

Figure 1: First eigenmode.

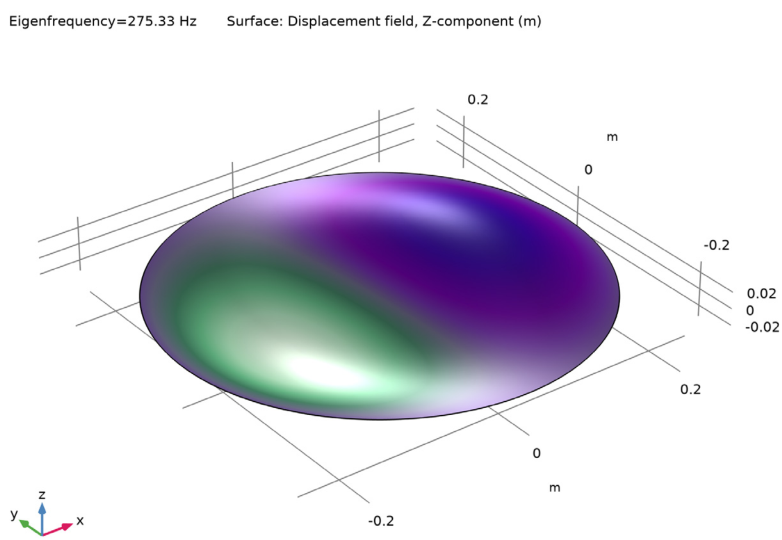

Figure 2: Second eigenmode.

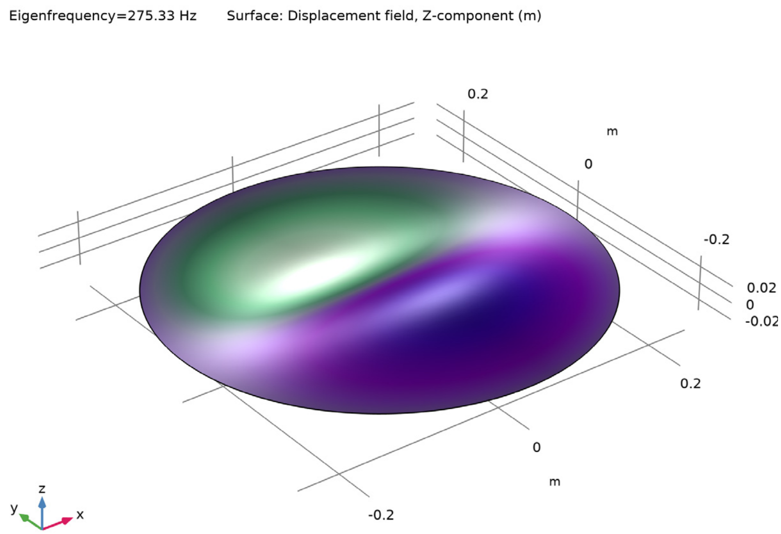

Figure 3: Third eigenmode.

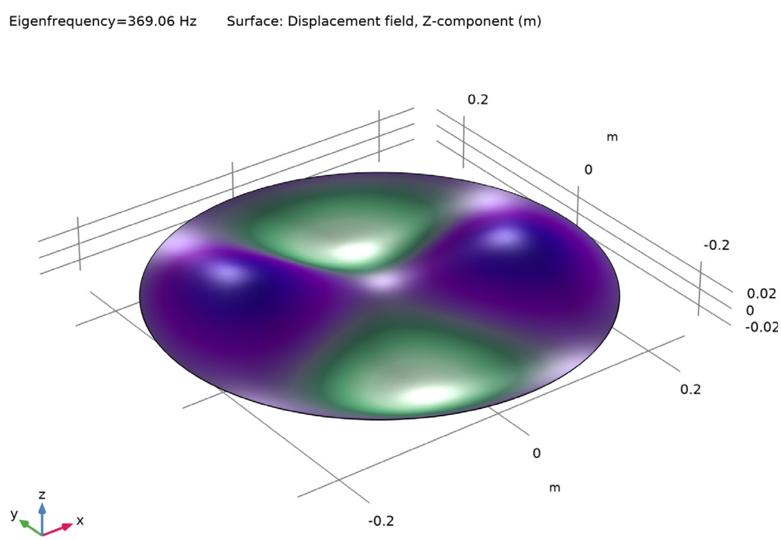

Figure 4: Fourth eigenmode.

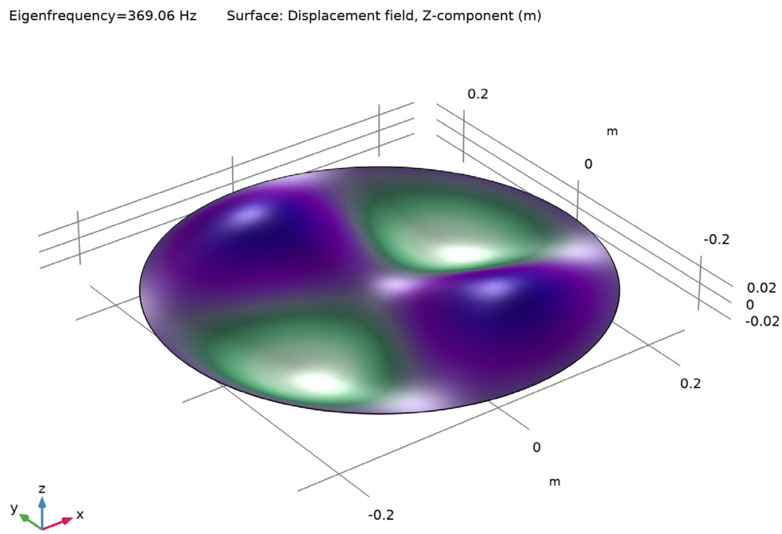

Figure 5: Fifth eigenmode.

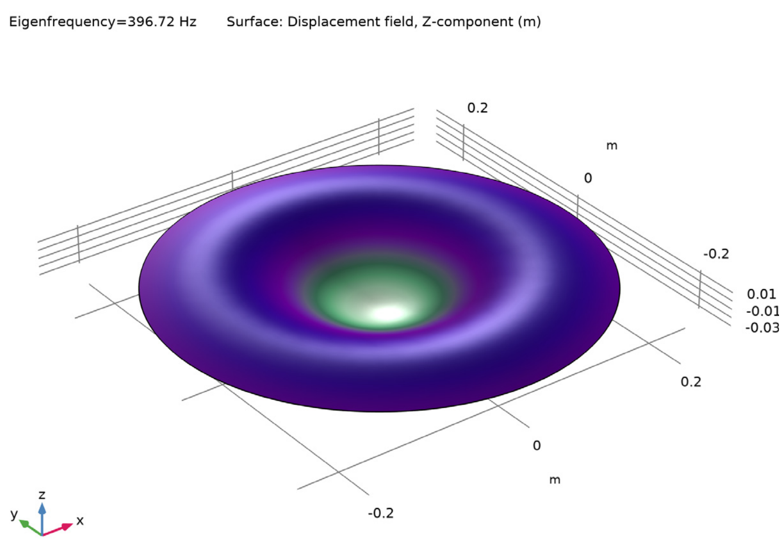

Figure 6: Sixth eigenmode.

Notes About the COMSOL Implementation

An eigenfrequency simulation with a pre-stressed structure can be simulated in two ways. If stresses are known in advance, it is possible to use an initial stress condition. This is shown in the first study.

In a general case, the prestress is given by some external loading, and is thus the result of a previous step in the solution. Such a study would consist of two steps: One stationary step for computing the prestressed state, and one step for the eigenfrequency. The special study type Prestressed Analysis, Eigenfrequency can be used to set up such a sequence. This is shown in the second study in this example.

Since an unstressed membrane has no stiffness in the transverse direction, it is generally difficult to get an analysis to converge without taking special measures. One such method is shown in the second study: A spring foundation is added during initial loading, and is then removed.

Reference

1. A. Bower, *Applied Mechanics of Solids*, CRC Press, 2010.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/vibrating_membrane`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Membrane (mbrn)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Eigenfrequency**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
R	250[mm]	0.25 m	Radius
thic	0.2[mm]	2E-4 m	Thickness
T0	100[MPa]*thic	20000 N/m	Pretension force
E1	200[GPa]	2E11 Pa	Young's modulus
rho1	7850[kg/m^3]	7850 kg/m³	Density
nu1	0.33	0.33	Poisson's ratio
fct	sqrt(T0/(thic* rho1))/(2*pi*R)	71.853 1/s	Common factor in natural frequencies
f10	2.4048*fct	172.79 1/s	1st natural frequency
f11	3.8317*fct	275.32 1/s	2nd and 3d natural frequencies
f12	5.1356*fct	369.01 1/s	4th and 5th natural frequencies
f20	5.5201*fct	396.64 1/s	6th natural frequency

DEFINITIONS

Cylindrical System 2 (sys2)

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

2 Right-click **Definitions** and choose **Coordinate Systems>Cylindrical System**.

GEOMETRY 1

Work Plane 1 (wp1)

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

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

3 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

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

1 In the **Work Plane** toolbar, click **Circle**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Radius** text field, type `R`.
4 In the **Model Builder** window, right-click **Geometry 1** and choose **Build All**.
5 Click the **Zoom Extents** button in the **Graphics** toolbar.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, locate the **Material Contents** section.
3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	E1	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	nu1	1	Young's modulus and Poisson's ratio
Density	rho	rho1	kg/m³	Basic

MEMBRANE (MBRN)

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Membrane (mbrn)** click **Thickness and Offset 1**.
2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.
3 In the d_0 text field, type `thic`.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Initial Stress and Strain 1

1 In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.
2 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

3 In the N_0 table, enter the following settings:

T0	0
0	T0

Prescribed Displacement 1

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

2 Select all four edges.

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

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

Fixed Constraint 1

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

2 Select Point 1 only.

Prescribed Displacement 2

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

2 Select Point 2 only.

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

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

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

STUDY 1

Step 1: Eigenfrequency

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

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

3 Find the **Values of linearization point** subsection. Select the **Include geometric nonlinearity** check box.

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

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Mode Shape (mbrn)** node, then click **Surface 1**.

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

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

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

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

Mode Shape (mbrn)

1 In the **Model Builder** window, click **Mode Shape (mbrn)**.

2 From the **Eigenfrequency** list, choose the first frequency at **275.3** Hz.

3 In the **Mode Shape (mbrn)** toolbar, click **Plot**.

4 From the **Eigenfrequency** list, choose the first frequency at **275.3** Hz.

5 In the **Mode Shape (mbrn)** toolbar, click **Plot**.

6 From the **Eigenfrequency** list, choose the first frequency at **369.1** Hz.

7 In the **Mode Shape (mbrn)** toolbar, click **Plot**.

8 From the **Eigenfrequency** list, choose the first frequency at **369.1** Hz.

9 In the **Mode Shape (mbrn)** toolbar, click **Plot**.

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

11 From the **Eigenfrequency (Hz)** list, choose **396.72**.

12 In the **Mode Shape (mbrn)** toolbar, click **Plot**.

Now, prepare a second study where the prestress is instead computed from an external load.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Eigenfrequency, Prestressed**.

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

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

MEMBRANE (MBRN)

Edge Load 1

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

2 Select all four edges.

3 In the **Settings** window for **Edge Load**, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

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

T0	r
0	phi
0	a

Add a spring with an arbitrary, small stiffness in order to suppress the out-of-plane singularity of the unstressed membrane.

Spring Foundation 1

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

2 Select Boundary 1 only.

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

4 From the list, choose **Diagonal**.

5 In the $\mathbf{k}_A$ table, enter the following settings:

0	0	0
0	0	0
0	0	10

Switch off the initial stress, which should not be part of the second study. In the eigenfrequency step, the stabilizing spring support must also be removed.

STUDY 2

Step 1: Stationary

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

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

3 Select the **Include geometric nonlinearity** check box.

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

5 In the tree, select **Component 1 (comp1)>Membrane (mbrn), Controls spatial frame>Linear Elastic Material 1>Initial Stress and Strain 1**.

6 Right-click and choose **Disable**.

Step 2: Eigenfrequency

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

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

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

4 In the tree, select **Component 1 (comp1)>Membrane (mbrn), Controls spatial frame>Linear Elastic Material 1>Initial Stress and Strain 1** and **Component 1 (comp1)>Membrane (mbrn), Controls spatial frame>Spring Foundation 1**.

5 Right-click and choose **Disable**.

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

RESULTS

Mode Shape (mbrn) 1

The eigenfrequencies computed using this more general approach are the same as before, except some small numerical differences.

To make **Study 1** behave as when it was first created, the features added for **Study 2** must be disabled.

STUDY 1

Solver Configurations

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

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

3 In the tree, select **Component 1 (comp1)>Membrane (mbrn), Controls spatial frame>Edge Load 1** and **Component 1 (comp1)>Membrane (mbrn), Controls spatial frame>Spring Foundation 1**.

4 Click **Disable**.