Vibrating Membrane

Introduction

In the following example you compute the natural frequencies of a pre-tensioned membrane using the 3D Membrane interface. This is an example of "stress stiffening"; in fact 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, $\rho = 7850$ kg/m^3

CONSTRAINTS

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

LOAD

The membrane is pre-tensioned by a homogenous biaxial stress of $\sigma_i = 100$ 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:

©2012 COMSOL

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

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

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

TABLE 1: COMPARISON BETWEEN ANALYTICAL AND COMPUTED NATURAL FREQUENCIES

Mode number	Factor	Analytical frequency (Hz)	COMSOL result (Hz)
1	k_{10} = 2.4048	172.8	172.8
2	k_{11} = 3.8317	275.3	275.3
3	k_{11} = 3.8317	275.3	275.3
4	k_{12} = 5.1356	369.0	369.1
5	k_{12} = 5.1356	369.0	369.1
6	k_{20} = 5.5201	396.6	396.7

 ©2012 COMSOL

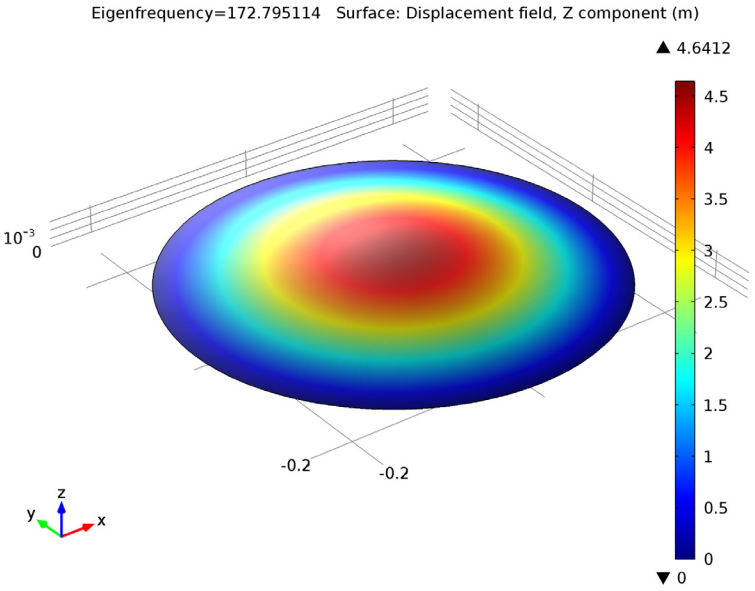

Figure 1: First eigenmode.

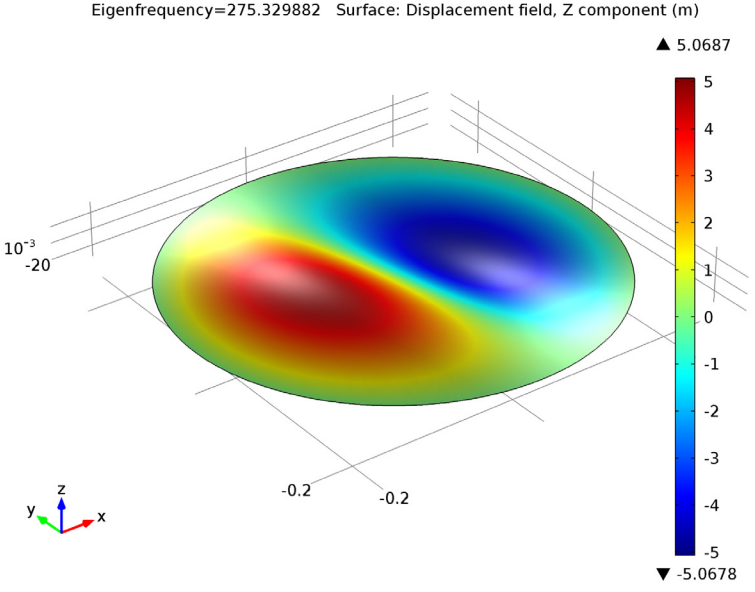

Figure 2: Second eigenmode.

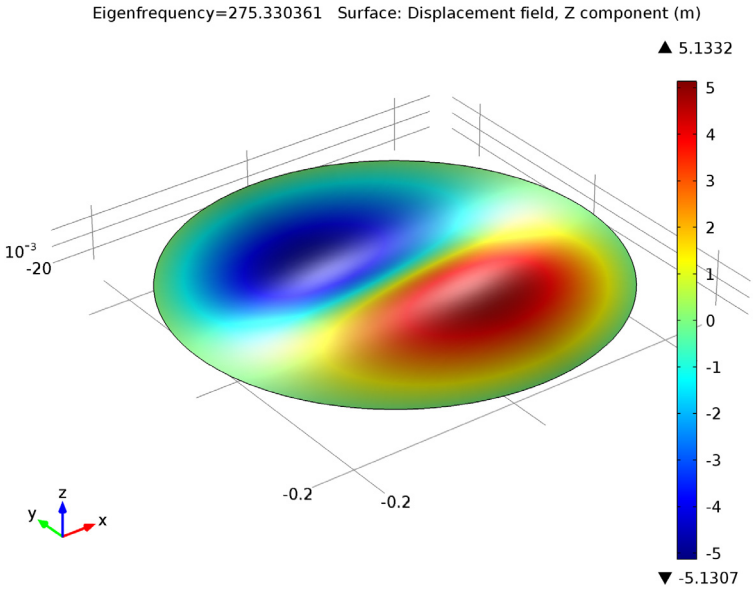

Figure 3: Third eigenmode.

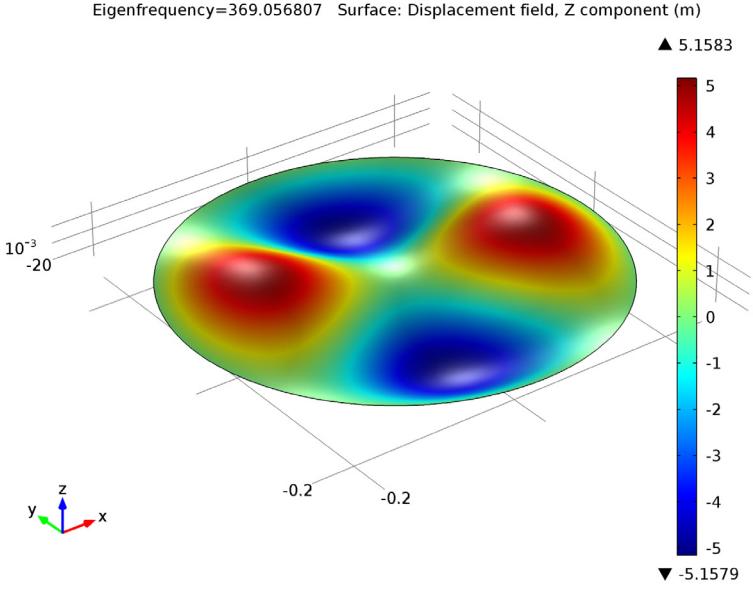

Figure 4: Fourth eigenmode.

 ©2012 COMSOL

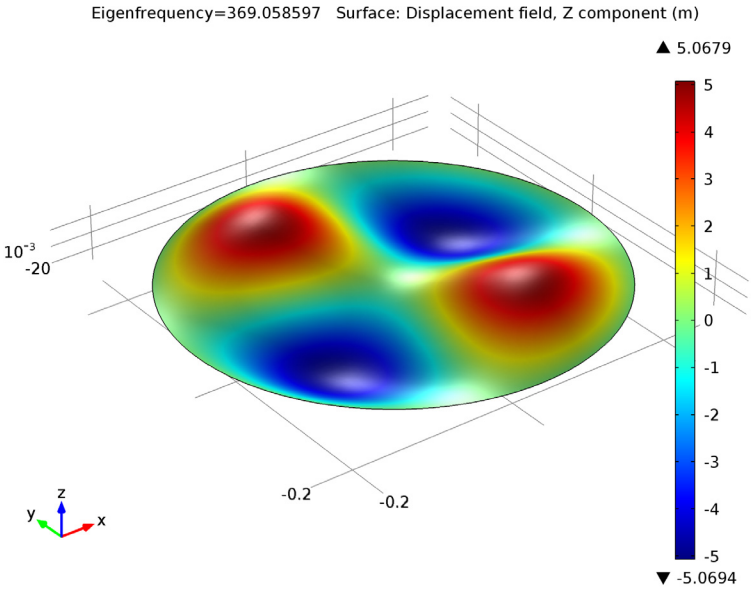

Figure 5: Fifth eigenmode.

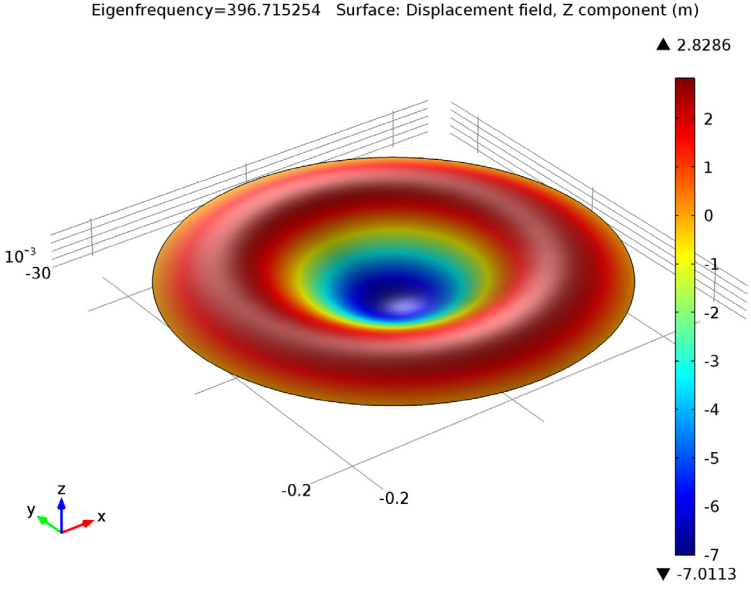

Figure 6: Sixth eigenmode.

Notes About the COMSOL Implementation

In this model the stresses are known in advance, so it is possible to use an initial stress condition. This is shown in the first study.

In a general case, the prestress is given by some external loading, 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 will generally be 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. Bower A., "Applied Mechanics of Solids", CRC Press, 2010.

Model Library path: `Structural_Mechanics_Module/Verification_Models/vibrating_membrane`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Structural Mechanics>Membrane (mem)**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Eigenfrequency**.

6 Click **Finish**.

GLOBAL DEFINITIONS

In the **Model Builder** window, expand the **Global Definitions** node.

 ©2012 COMSOL

Parameters

1 Right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

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

DEFINITIONS

1 In the **Model Builder** window, expand the **Model 1>Definitions** node.

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

GEOMETRY 1

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

Circle 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Circle**.

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

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

MATERIALS

Material 1

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value
Young's modulus	E	E1
Poisson's ratio	nu	nu1
Density	rho	rho1

MEMBRANE

1 In the **Model Builder** window, under **Model 1** click **Membrane**.

2 In the **Membrane** settings window, locate the **Thickness** section.

3 In the *d* edit field, type `thic`.

Initial Stress and Strain 1

1 In the **Model Builder** window, under **Model 1>Membrane** right-click **Linear Elastic Material 1** and choose **Initial Stress and Strain**.

2 In the **Initial Stress and Strain** settings window, 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 **Model Builder** window, right-click **Membrane** and choose **Prescribed Displacement**.

2 Select all four edges.

3 In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.

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

Fixed Constraint 1

1 Right-click **Membrane** and choose **Points>Fixed Constraint**.

2 Select Point 1 only.

Prescribed Displacement 2

1 Right-click **Membrane** and choose **Points>Prescribed Displacement**.

2 Select Point 2 only.

 ©2012 COMSOL

3 In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.

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

MESH 1

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

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

STUDY 1

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

RESULTS

Mode shape

1 In the **Model Builder** window, expand the **Mode shape** node, then click **Surface**.

2 In the **Surface** settings window, locate the **Expression** section.

3 In the **Expression** edit field, type `w`.

4 Click the **Plot** button.

5 In the **Model Builder** window, click **Mode shape**.

6 In the **3D Plot Group** settings window, locate the **Data** section.

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

8 Click the **Plot** button.

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

10 From the **Eigenfrequency** list, choose the second frequency at **275.3** Hz.

11 Click the **Plot** button.

12 In the **3D Plot Group** settings window, locate the **Data** section.

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

14 Click the **Plot** button.

15 In the **3D Plot Group** settings window, locate the **Data** section.

16 From the **Eigenfrequency** list, choose the second frequency at **369.1** Hz.

17 Click the **Plot** button.

18 In the **3D Plot Group** settings window, locate the **Data** section.

19 From the **Eigenfrequency** list, choose **396.715254**.

©2012 COMSOL

20 Click the **Plot** button.

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

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Find the **Studies** subsection. In the tree, select **Preset Studies>Prestressed Analysis, Eigenfrequency**.

3 Click **Finish**.

STUDY 2

MEMBRANE

Edge Load 1

1 In the **Model Builder** window, under **Model 1** right-click **Membrane** and choose **Edge Load**.

2 Select all four edges

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

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

5 Locate the **Force** section. In the $\mathbf{F}_\mathrm{L}$ table, enter the following settings:

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 **Model Builder** window, right-click **Membrane** and choose **Spring Foundation**.

2 Select Boundary 1 only.

3 In the **Spring Foundation** settings window, locate the **Spring** section.

 ©2012 COMSOL

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

0	x
0	y
10	z

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 **Stationary** settings window, locate the **Physics and Variables Selection** section.

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

4 In the **Physics and variables selection** tree, select **Model 1>Membrane>Linear Elastic Material 1>Initial Stress and Strain 1**.

5 Click **Disable**.

Step 2: Eigenfrequency

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

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

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

4 In the **Physics and variables selection** tree, select **Model 1>Membrane>Linear Elastic Material 1>Initial Stress and Strain 1** and **Model 1>Membrane>Spring Foundation 1**.

5 Click **Disable**.

6 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

RESULTS

Mode shape 1

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

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

STUDY 1

Step 1: Eigenfrequency

1. In the **Eigenfrequency** settings window, locate the **Physics and Variables Selection** section.
2. Select the **Modify physics tree and variables for study step** check box.
3. In the **Physics and variables selection** tree, select **Model 1>Membrane>Edge Load 1** and **Model 1>Membrane>Spring Foundation 1**.
4. Click **Disable**.

 ©2012 COMSOL