Model created in COMSOL Multiphysics 6.4

Nonlinear Harmonic Response

Introduction

This tutorial model shows how to evaluate the harmonic response of a structure with a moderately nonlinear behavior. To solve such a nonlinear problem accurately, it is necessary to use a time-domain analysis. Solving in the time domain can, however, be time consuming as it requires several periods until a steady-state solution is reached. A way to speed up such an analysis is to use a linearized frequency-response analysis to provide good initial conditions. This approach is described in this tutorial.

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.

LOAD

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

A harmonically varying pressure with a 50 Hz frequency and a magnitude of $F_0 = 10$ kPa is applied on the membrane.

Results and Discussion

Figure 1 shows the vertical displacement at the disk center. The peak displacement is about 6.8 mm, which is significantly less than for the linearized solution (about 8.6 mm). The steady state is reached within a few periods, thanks to the initial condition using the linearized frequency-domain solution. This way, the analysis time is significantly reduced.

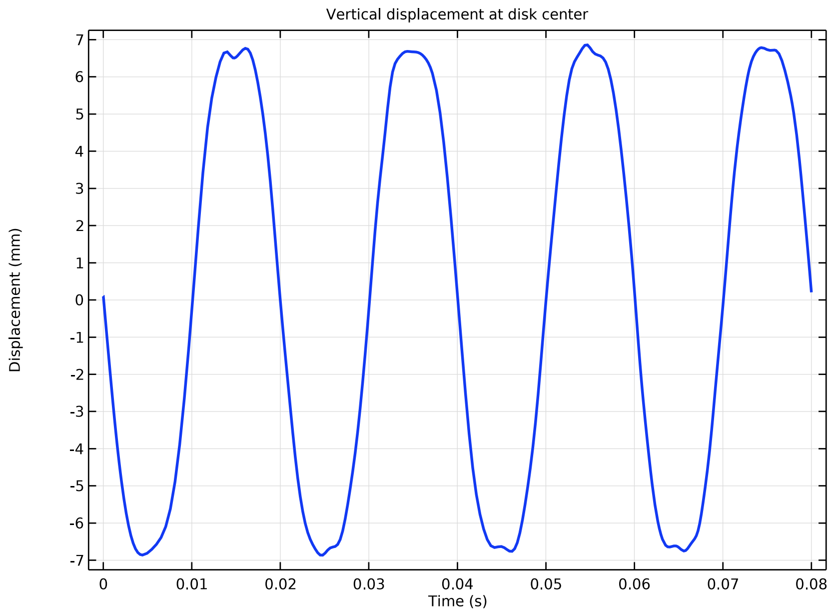

Figure 1: Vertical displacement of the disk center versus time near steady state.

For comparison, Figure 2 shows the same vertical displacement at the disk center, but this time computed using the default initial condition. While the same peak displacement value is obtained, it requires more than 10 periods to reach the steady state.

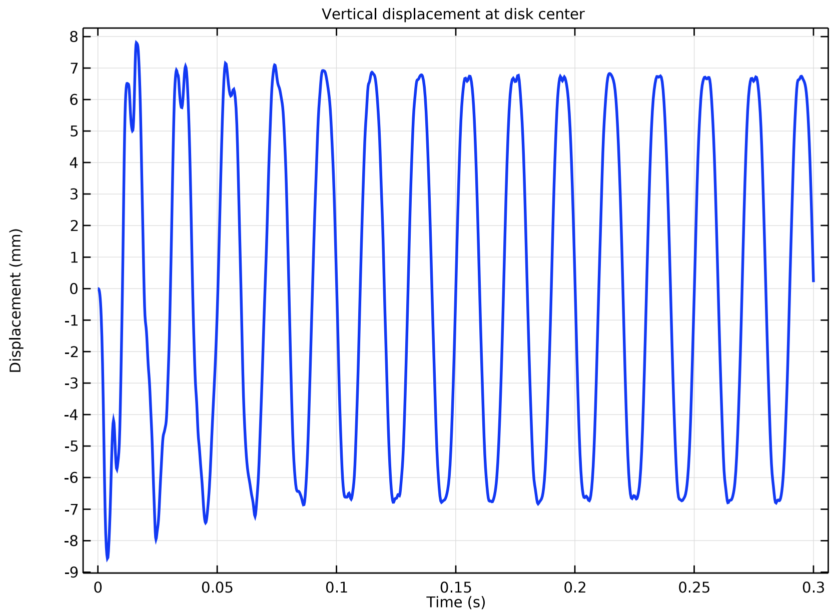

Figure 2: Vertical displacement of the disk center versus time (computed from zero displacement/velocity).

For comparison, Figure 3 shows the displacement magnitude at the center of disk computed with both the linear harmonic and the transient studies. The blue line corresponds to the linear solution computed with the linear harmonic study, while the green line is the solution including geometric nonlinearity effect that can only be consider in the transient study.

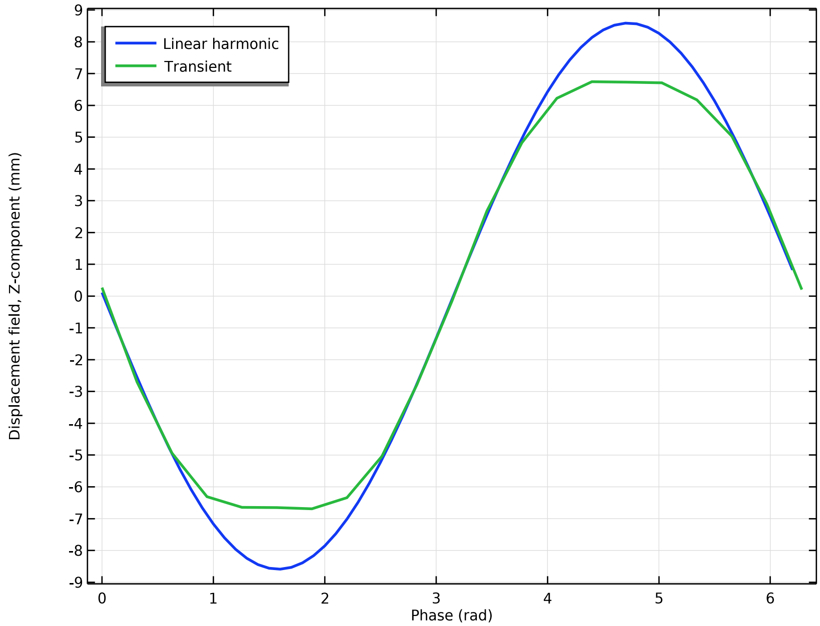

Figure 3: Displacement at the disk center computed with linear harmonic study (blue) vs transient study (green).

In Figure 4 the displacement at t = 0.075 s, corresponding to the peak deformation of the membrane, is shown.

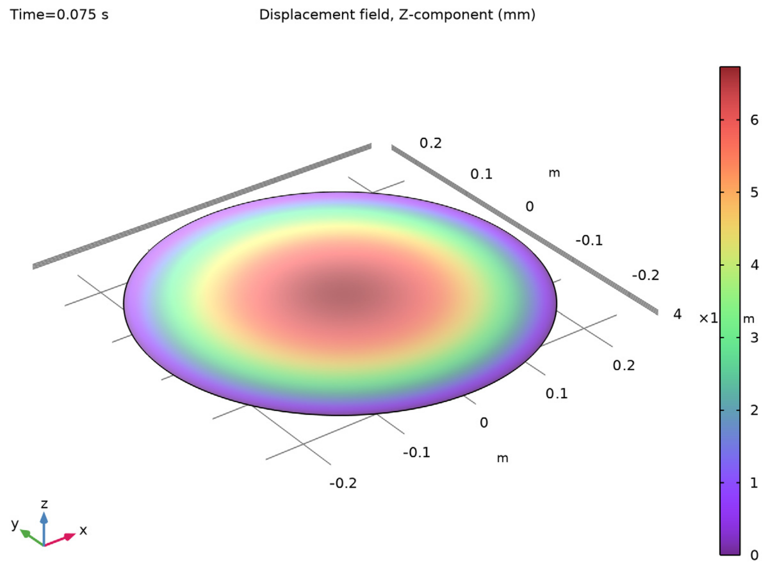

Figure 4: Vertical displacement from time domain solution.

For comparison, Figure 5 shows the displacement from the frequency domain solution.

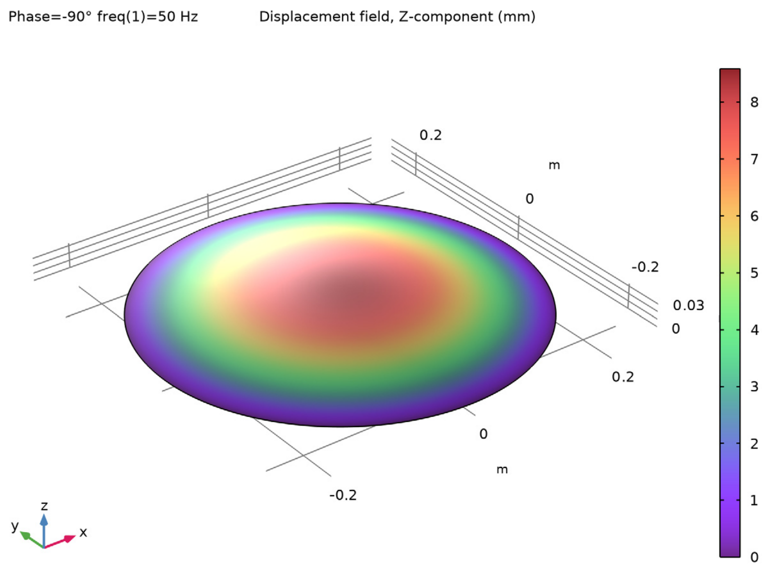

Figure 5: Vertical displacement from frequency domain solution.

Notes About the COMSOL Implementation

To save computational time to reach a steady-state solution using a periodic time-domain analysis, the initial conditions must be carefully selected. For a structural dynamics problem you need to set both the displacement and velocity fields at the initial time. The solution from a linearized harmonic response is a good choice and easy to obtain.

In COMSOL Multiphysics, you can use the `withsol()` operator to evaluate expression from a different solution. The first argument of the operator has to be the tag of the solution to use and the second argument the expression to evaluate.

Table 1 shows the expressions used as the initial condition for the displacement field.

TABLE 1: INITIAL CONDITIONS FOR THE DISPLACEMENT FIELD.

`withsol('sol1',real(u))`	X
`withsol('sol1',real(u))`	Y
`withsol('sol1',real(u))`	Z

Similarly, Table 2 shows the initial structural velocity field.

TABLE 2: INITIAL CONDITIONS FOR THE STRUCTURAL VELOCITY FIELD.

`withsol('sol1',real(mbrn.u_tX))`	X
`withsol('sol1',real(mbrn.u_tY))`	Y
`withsol('sol1',real(mbrn.u_tZ))`	Z

Application Library path: `Structural_Mechanics_Module/Tutorials/nonlinear_harmonic`

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 **Preset Studies for Selected Physics Interfaces** > **Frequency Domain, Prestressed**.

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
th	0.2[mm]	2E-4 m	Thickness

Name	Expression	Value	Description
T0	100[MPa]*th	20000 N/m	Pretension force
F0	10[kPa]	10000 Pa	Excitation load magnitude
freq0	50[Hz]	50 Hz	Excitation 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 **Go to Plane Geometry**.

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.

Work Plane 1 (wp1) > Point 1 (pt1)

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

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

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

ADD MATERIAL

1 In the **Materials** 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 Right-click and choose **Add to Component 1 (comp1)**.

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

MEMBRANE (MBRN)

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 `170`.
5 In the ζ_1 text field, type `2e-2`.
6 In the f_2 text field, type `400`.
7 In the ζ_2 text field, type `4e-2`.

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 `th`.

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement**.
2 In the **Settings** window for **Prescribed Displacement**, locate the **Edge Selection** section.
3 From the **Selection** list, choose **All edges**.
4 Locate the **Prescribed Displacement** section. From the **Displacement in z direction** list, choose **Prescribed**.

To avoid rigid body motion add a **Rigid Motion Suppression** feature.

Rigid Motion Suppression 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Rigid Motion Suppression**.
2 In the **Settings** window for **Rigid Motion Suppression**, locate the **Contributing Points** section.
3 From the list, choose **From selection input**.
4 Locate the **Point Selection** section. Click to select the **Activate Selection** toggle button.
5 Select Points 2, 4, and 5 only.

Edge Load 1

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

2 In the **Settings** window for **Edge Load**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **All edges**.

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

5 Locate the **Force** section. Specify the $\mathbf{f}_\mathrm{L}$ 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 Specify the $\mathbf{k}_\mathrm{A}$ matrix as

0	0	0
0	0	0
0	0	10

Face Load 1

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

2 Select Boundary 1 only.

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

4 From the **Load type** list, choose **Pressure**.

5 In the p text field, type F0.

6 Right-click **Face Load 1** and choose **Harmonic Perturbation**.

In a frequency domain analysis, the zero phase corresponds to cos(time). Change the excitation load phase so that it represents sin(time) since such a load will be used in the time-domain analysis.

Phase 1

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

2 In the **Settings** window for **Phase**, locate the **Load Phase** section.

3 In the ϕ text field, type `-pi/2`.

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 **Coarser**.

STUDY 1

Step 2: Frequency-Domain Perturbation

1 In the **Model Builder** window, under **Study 1** click **Step 2: Frequency-Domain Perturbation**.

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

3 In the **Frequencies** text field, type `freq0`.

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

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

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

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

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

9 Clear the **Generate default plots** checkbox.

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

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Membrane** > **Displacement (mbrn)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Displacement - Frequency Domain

1. In the **Settings** window for **3D Plot Group**, type `Displacement - Frequency Domain` in the **Label** text field.
2. Locate the **Phase** section. From the **Solution at angle (phase)** list, choose **Manual**.
3. In the **Phase** text field, type `-90`.
4. In the **Model Builder** window, expand the **Results** > **Datasets** node.

Surface 1

1. In the **Model Builder** window, expand the **Displacement - Frequency Domain** 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. From the **Unit** list, choose **mm**.
5. In the **Displacement - Frequency Domain** toolbar, click **Plot**.

Point Evaluation 1

1. In the **Results** toolbar, click 8.85 e-12 **Point Evaluation**.
2. Select Point 3 only.
3. In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.
4. In the table, enter the following expression and unit:

Expression	Unit	Description
`w`	`mm`	`Displacement field, Z-component`

5. From the **Expression evaluated for** list, choose **Peak value for total solution**.
6. Click **Evaluate**.

MEMBRANE (MBRN)

The next step is to set initial conditions using the solution computed by the frequency response analysis.

Initial Values 2

1. In the **Physics** toolbar, click **Boundaries** and choose **Initial Values**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

4 Specify the **u** vector as

`withsol('sol1',real(u))`	X
`withsol('sol1',real(v))`	Y
`withsol('sol1',real(w))`	Z

5 Specify the Structural velocity field as:

withsol(’sol1’,real(mbrn.u_tX))	X
withsol(’sol1’,real(mbrn.u_tY))	Y
withsol(’sol1’,real(mbrn.u_tZ))	Z

Face Load 2

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

2 Select Boundary 1 only.

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

4 From the **Load type** list, choose **Pressure**.

5 In the *p* text field, type `sin(2*pi*t*freq0)*F0`.

DEFINITIONS

Point Probe 1 (point1)

1 In the **Definitions** toolbar, click **Probes** and choose **Point Probe**.

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

3 Click **Clear Selection**.

4 Select Point 3 only.

5 Locate the **Expression** section. In the **Expression** text field, type `w`.

6 From the **Table and plot unit** list, choose **mm**.

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 Right-click and choose **Add Study**.

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

STUDY 2

Step 1: Time Dependent

With good initial conditions you only need to compute for a few periods to reach steady state.

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
2 In the **Output times** text field, type `0 range(3,1/20,4)/freq0`.
3 Select the **Include geometric nonlinearity** checkbox.
4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.
5 In the tree, select **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Spring Foundation 1**.
6 Right-click and choose **Disable**.
7 In the tree, select **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Face Load 1**.
8 Right-click and choose **Disable**.
9 In the **Model Builder** window, click **Study 2**.
10 In the **Settings** window for **Study**, locate the **Study Settings** section.
11 Clear the **Generate default plots** checkbox.
12 In the **Study** toolbar, click **Compute**.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.
2 Go to the **Result Templates** window.
3 In the tree, select **Study 2/Solution 3 (sol3)** > **Membrane** > **Displacement (mbrn)**.
4 Click the **Add Result Template** button in the window toolbar.
5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Displacement - Time Domain

1 In the **Settings** window for **3D Plot Group**, type `Displacement - Time Domain` in the **Label** text field.
2 Locate the **Data** section. From the **Time (s)** list, choose **0.075**.

Surface 1

1 In the **Model Builder** window, expand the **Displacement - Time Domain** 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 From the **Unit** list, choose **mm**.

5 In the **Displacement - Time Domain** toolbar, click **Plot**.

Displacement

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 2**.

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

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

4 In the **Title** text area, type `Vertical displacement at disk center`.

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** checkbox. In the associated text field, type `Displacement (mm)`.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Displacement** node, then click **Probe Table Graph 1**.

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

3 From the **Width** list, choose **2**.

4 Click to expand the **Legends** section. Clear the **Show legends** checkbox.

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

Linear Harmonic vs. Transient

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

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

3 From the **Title type** list, choose **None**.

4 In the **Label** text field, type `Linear Harmonic vs. Transient`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `Phase (rad)`.

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

Point Graph 1

1 Right-click **Linear Harmonic vs. Transient** and choose **Point Graph**.

2 Select Point 3 only.

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

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

5 From the **Unit** list, choose **mm**.

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

7 In the **Phase** text field, type `range(0,0.1,2*pi)`.

8 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

9 Click to expand the **Legends** section. Select the **Show legends** checkbox.

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

11 In the table, enter the following settings:

Legends
Linear harmonic

Point Graph 2

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

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

3 From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.

4 From the **Time selection** list, choose **Interpolated**.

5 In the **Times (s)** text field, type `range(6e-2,1e-3,8e-2)`.

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

7 In the **Expression** text field, type `(t-0.06)*2*pi*freq0`.

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

Legends
Transient

9 In the **Linear Harmonic vs. Transient** toolbar, click **Plot**.

The modeling part is now finished. Follow the steps below if you want to run the model later with modified parameter values.

STUDY 1

Step 1: Stationary

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

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

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Face Load 1**, **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Initial Values 2**, and **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Face Load 2**.

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

Step 2: Frequency-Domain Perturbation

1 In the **Model Builder** window, click **Step 2: Frequency-Domain Perturbation**.

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

3 In the tree, select **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Initial Values 2**.

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

5 In the tree, select **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Face Load 2**.

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