Axisymmetric Condenser Microphone

Introduction

This is a model of a condenser microphone with a simple axisymmetric geometry. The model aims to give a precise description of the physical working principles of such a microphone.

The condenser microphone is considered to be the microphone with highest quality when performing precise acoustical measurements and with high-fidelity reproduction properties when performing sound recordings; see Ref. 2. This electro-mechanical acoustic transducer works by transforming the mechanical deformation of a thin membrane (diaphragm) into an AC voltage signal.

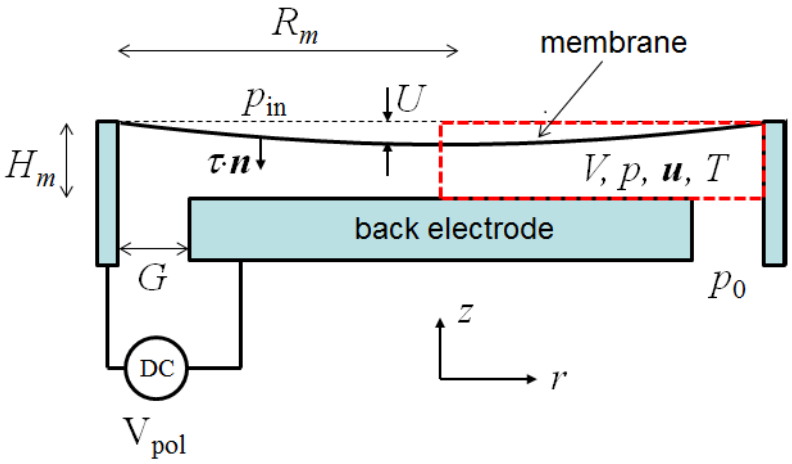


Figure 1: Sketch of the condenser microphone system including variables and coordinate system. The red box indicates the modeled region.

Models for describing condenser microphones have classically been of the equivalent network type; see Ref. 2. Analytical models exist for simpler geometries, but there are also highly advanced analytical models for more complex geometries; see for example Ref. 1 or Ref. 4. In the present detailed finite-element model the thermoacoustic, electric, and structural problem is solved fully coupled using the frequency-domain linear perturbation solver. This includes the DC charging (pre-polarization) and deformation of the membrane which makes out the zeroth order linearization point. The system analyzed here is identical to that of the Axisymmetric Condenser

Microphone with Electrical Lumping model except that it does not use any lumping approximations. In many cases, lumping of the electrical part is a good approximation, especially for simple geometries such as this one, where the back electrode is flat and has no perforations. During the initial design steps, lumped models are an important tool. In more complex geometries, solving the full set of equations is necessary to get the correct response.

The current model represents a true multiphysics problem that involves several physics interfaces: Thermoacoustics, Electrostatics, Moving Mesh, and a Membrane model.

Note: This model requires the AC/DC and the Structural Mechanics Module as well as the Acoustics Module.

Model Definition

The geometry and model definitions are depicted in Figure 1. In many microphones there is a back-volume below the electrode. In this model a simplified approach is taken and a pressure release condition is applied with p_0 = 0 Pa. The membrane is deformed due to the electrostatic forces from charging the capacitor and because of the pressure variation from the external incoming uniform acoustic signal p_{in}. The chosen dimensions of the microphone are typical generic dimensions. Dimensions and parameters are given in Table 1.

TABLE 1: MICROPHONE DIMENSIONS AND PARAMETERS

SYMBOL	SIZE & UNIT	DESCRIPTION
H_{m}	18 µm	Air gap thickness
R_{m}	2 mm	Membrane radius
G	54 µm	Slit gap width
T_{m0}	3150 N/m	Membrane static tension
E_{m}	221 GPa	Membrane elastic modulus
t_{m}	7 µm	Membrane thickness
ρ_{m}	8300 kg/m^3	Membrane density
V_{pol}	100 V	Target polarization voltage
Q_{m}	$6\cdot10^{-10}$ C	Static polarization charge
ν_{m}	0.4	Poisson's ratio for the membrane

The membrane is backed by a thin air gap of thickness H_{m} and a back electrode. Because the gap is so small, the inclusion of thermal and viscous losses in the acoustic model is essential, thus using the thermoacoustic interface. The membrane and back electrode make up a capacitor that is polarized by an external DC voltage source, which equals a surface charge Q_{m}. The air gap acts as a damping layer for the membrane vibrations. As the gap between the membrane and the back electrode varies, a voltage change is induced. This AC voltage is the output of the microphone.

The sensitivity of the condenser microphone, L, is measured in the unit dB (relative to 1 V/Pa). It is defined as the ratio of the open circuit output voltage V_{out} to the input pressure p_{in} and is given by

$$L = 20\log\left[\left|\frac{V_{\mathrm{out}}}{p_{\mathrm{in}}}\right| / \left(1\frac{\mathrm{V}}{\mathrm{Pa}}\right)\right] \quad (1)$$

In the present model, the membrane (or diaphragm) is modeled using the dedicated Membrane interface from the Structural Mechanics Module. The membrane is subject to a surface load that is the sum of the external incident pressure p_{in}, the internal pressure $p = p(\mathbf{r})$ (given by the thermoacoustic model), and the electrostatic force given by the Maxwell surface stress, $\mathbf{n} \cdot \boldsymbol{\tau}$. The incident pressure is here assumed to be uniform over the microphone membrane, which is only an approximation. At the highest frequencies modeled, the acoustic wavelength becomes comparable with the membrane radius.

Results and Discussion

This model involves a detailed description of the physical effects in a simple condenser microphone. The sensitivity of the microphone, L, is directly determined from the model (voltage on the terminal divided by the incident pressure) and is depicted in Figure 2 below.

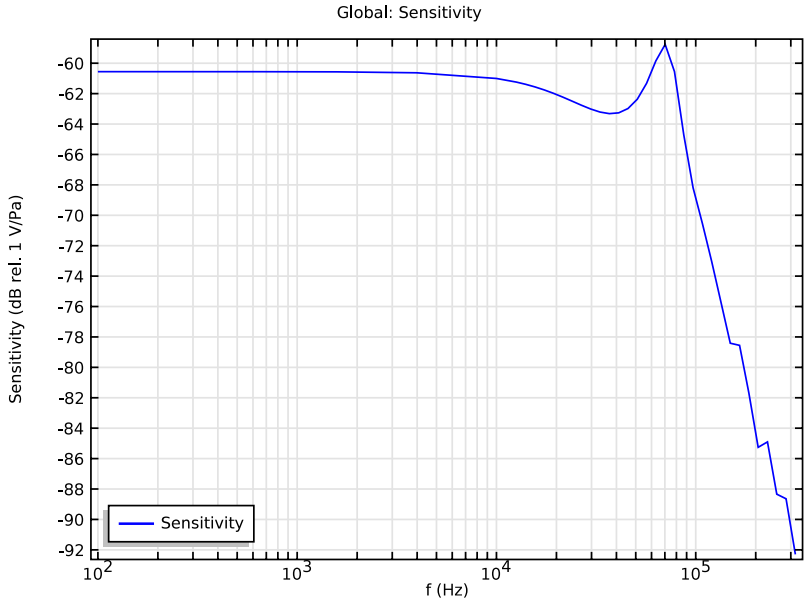


Figure 2: Sensitivity curve of the microphone measured in dB relative to 1 V/Pa.

The shape of the deformed membrane is plotted for $f = 0.3$ GHz as a 3D surface in Figure 3, using a revolution 2D data set. At this frequency it is clear to see how higher order modes in the membrane are the cause for the poor sensitivity.

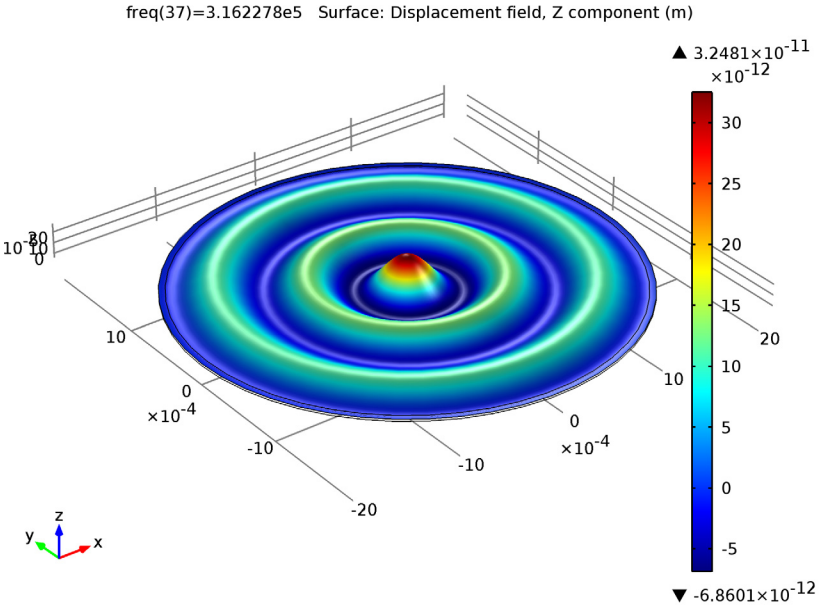


Figure 3: 3D representation of the harmonic membrane deformation at 0.3 GHz.

The principles described in this model can be extended to 3D models with more complex geometries. Because the full set of equations is solved, such a model includes all physical effects to a high degree of detail. It can be used to optimize the performance of microphones, to make virtual tests of new geometries, or to investigate the relative importance of different parameters.

Notes About the COMSOL Implementation

COUPLED STATIC AND FREQUENCY-DOMAIN MODEL USING THE LINEAR PERTURBATION SOLVER

The current model solves a fully coupled problem using the linear perturbation frequency-domain solver. A stationary model determines the linearization point and the full system of equations is then linearized and solved around this point to determine the harmonic small-signal response.

The first step is to determine the linearization point for the problem which requires solving a static model that determines the shape of the membrane after the polarization

voltage is applied. The current model does not consider the transient charging of the condenser. The first step solves the electrostatic model (using the Electrostatics interface of the AC/DC Module) coupled to the membrane model. The acoustic model is deactivated as it is a small perturbation and thus has no contribution to the static solver step. To determine the correct capacitance, C_0, a Moving Mesh interface is needed, because the capacitance is a geometry-dependent quantity.

The second step is to solve the linear perturbation frequency-domain model that describes the time-harmonic small-signal deformation of the membrane and the interaction with the fluid (described by a Thermoacoustics, Frequency Domain interface) within the microphone.

The coupling between the acoustic and mechanical problems is achieved by imposing a constraint on the membrane deformation, $\mathbf{u}_\mathrm{m}$ (SI unit: m), and the acoustic velocity, $\mathbf{u}$ (SI unit: m/s):

$$\mathbf{u} = i\omega\mathbf{u}_\mathrm{m} \tag{2}$$

where the multiplication with $i\omega$ corresponds to a time derivative in frequency domain. Because constraints are bidirectional in COMSOL Multiphysics, this is a two-way coupling.

References

1. T. Lavergne, S. Durand, M. Bruneau, N. Joly, and D. Rodrigues, "Dynamic behavior of the circular membrane of an electrostatic microphone: Effect of holes in the backing electrode," *J. Acoust. Soc. Am.*, vol. 128, p. 3459, 2010.

2. W. Marshall Leach, Jr., *Introduction to Electroacoustics and Audio Amplifier Design*, 3rd ed., Kendall/Hunt Publishing Company, 2003.

3. P.M. Morse and K. Uno Ignard, *Theoretical Acoustics*, Princeton University Press, 1968.

4. V.C. Henriquez, *Numerical Transducer Modelling*, PhD Thesis, DTU, November 2001.

Model Library path: `Acoustics_Module/Industrial_Models/condenser_microphone`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click the **2D Axisymmetric** button.

2 In the **Select physics** tree, select **AC/DC>Electrostatics (es)**.

3 Click the **Add** button.

This is an appropriate choice because it is a good assumption that the electric processes in this model are quasistatic.

4 In the **Select physics** tree, select **Acoustics>Thermoacoustics>Thermoacoustics, Frequency Domain (ta)**.

5 Click the **Add** button.

6 In the **Select physics** tree, select **Structural Mechanics>Membrane (mem)**.

7 Click the **Add** button.

8 In the **Displacement field** edit field, type `um`.

9 In the **Displacement field components** table, enter the following settings:

um
vm
wm

The displacement field of the membrane is now (um,vm,wm) while the velocity field in the fluid is (u,v,w).

10 In the **Select physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.

11 Click the **Add** button.

12 Click the **Study** button.

13 In the tree, select **Custom Studies>Empty Study**.

14 Click the **Done** button.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file `condenser_microphone_parameters.txt`.

These are the parameters specifying the geometry and the physical properties of the microphone and membrane.

GEOMETRY 1

Rectangle 1

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

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `Rm`.

4 In the **Height** edit field, type `Hm`.

5 Click the **Build Selected** button.

Rectangle 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `G`.

4 In the **Height** edit field, type `Hm`.

5 Locate the **Position** section. In the ***r*** edit field, type `Rm-G`.

6 Click the **Build All Objects** button.

DEFINITIONS

Create a selection corresponding to the membrane for use when adding features to the membrane edge.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Explicit** settings window, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 3 and 6 only.

5 Right-click **Component 1>Definitions>Explicit 1** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Membrane` in the **New name** edit field.

7 Click **OK**.

MATERIALS

On the **Home** toolbar, click **Add Material**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Air**.

3 In the **Add material** window, click **Add to Component**.

MATERIALS

Air

1 In the **Model Builder** window, under **Component 1>Materials** click **Air**.

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Bulk viscosity	muB	muB0	Pa·s	Basic

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

Having set up the materials, proceed to setting up and defining the physics. Begin with the electric problem.

ELECTROSTATICS

1 In the **Model Builder** window, under **Component 1** click **Electrostatics**.

2 In the **Electrostatics** settings window, click to expand the **Equation** section.

3 From the **Equation form** list, choose **Stationary**.

Terminal 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**.

2 Select Boundary 2 only.

3 In the **Terminal** settings window, locate the **Terminal** section.

4 In the Q_0 edit field, type `Qm`.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 In the **Ground** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Membrane**.

Next, set up the acoustics before turning to the membrane model and the moving mesh interface.

THERMOACOUSTICS, FREQUENCY DOMAIN

Thermoacoustics Model 1

1 In the **Model Builder** window, under **Component 1>Thermoacoustics, Frequency Domain** click **Thermoacoustics Model 1**.

2 In the **Thermoacoustics Model** settings window, locate the **Thermoacoustics Model** section.

3 From the $\rho_0(p_0,T_0)$ list, choose **From material**.

Pressure (Adiabatic) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Pressure (Adiabatic)**.

2 Select Boundary 5 only.

3 In the **Pressure (Adiabatic)** settings window, locate the **Pressure** section.

4 In the p_{bnd} edit field, type `p0`.

Velocity 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Velocity**.

2 In the **Velocity** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Membrane**.

4 Locate the **Velocity** section. Select the **Prescribed in r direction** check box.

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

6 In the u_0 edit field, type `ta.iomega*um`.

7 In the w_0 edit field, type `ta.iomega*wm`.

The last step sets the acoustic velocity equal to the deformation of the membrane. This condition is a bidirectional constraint, meaning that a reaction force is added to the membrane equation, ensuring a two-way coupling.

Now, set up the membrane model, constrain it at the outer perimeter where it is fixed and add the forces that act on it. They are the electro static forces given by the Maxwell stress tensor (`es.dnTer,es.dnTez`) and the incident pressure field `pin`. Use the `linper()` operator to tell COMSOL that the incident pressure is only a harmonic frequency dependent quantity (not a static load).

MEMBRANE

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

2 In the **Membrane** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Membrane**.

4 Locate the **Thickness** section. In the d edit field, type `tm`.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1>Membrane** click **Linear Elastic Material 1**.

2 In the **Linear Elastic Material** settings window, locate the **Linear Elastic Material** section.

3 From the E list, choose **User defined**. In the associated edit field, type `Em`.

4 From the ν list, choose **User defined**. In the associated edit field, type `num`.

5 From the ρ list, choose **User defined**. In the associated edit field, type `rhom`.

Initial Stress and Strain 1

1 Right-click **Component 1>Membrane>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:

TmO	0
0	TmO

Fixed Constraint 1

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

2 Select Point 6 only.

Face Load 1

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

2 In the **Face Load** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Membrane**.

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

es.dnTer	r
es.dnTez	z

Face Load 2

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

2 In the **Face Load** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Membrane**.

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

5 In the p edit field, type `linper(pin)`.

MOVING MESH

Free Deformation 1

1 On the **Physics** toolbar, click **Domains** and choose **Free Deformation**.

2 Select Domains 1 and 2 only.

Prescribed Mesh Displacement 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 In the **Prescribed Mesh Displacement** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Membrane**.

4 Locate the **Prescribed Mesh Displacement** section. In the d_r edit field, type `um`.

5 In the d_z edit field, type `wm`.

Prescribed Mesh Displacement 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 Select Boundary 1 only.

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

4 Clear the **Prescribed z displacement** check box.

The above Moving Mesh interface ensures that the computational mesh deforms according to the membrane deformation (um,wm).

MESH 1

The final step before setting up the solvers is to build the computational mesh.

Mapped 1

In the **Model Builder** window, under **Component 1** right-click **Mesh 1** and choose **Mapped**.

Distribution 1

1 In the **Model Builder** window, under **Component 1>Mesh 1** right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 2 and 3 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type `30`.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 1 and 7 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** edit field, type `10`.

6 In the **Element ratio** edit field, type `2`.

7 Select the **Symmetric distribution** check box.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 5 and 6 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type `4`.

5 Click the **Build Selected** button.

Boundary Layer Properties

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Boundary Layers**.

2 Select Boundaries 4 and 7 only.

3 In the **Boundary Layer Properties** settings window, locate the **Boundary Layer Properties** section.

4 In the **Number of boundary layers** edit field, type `6`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** edit field, type `1e-6`.

7 Click the **Build All** button.

STUDY 1

Some manual work is required to set up the solvers. First, a stationary solver determines the linearization point for the frequency-domain perturbation solver. For this step, disable the acoustic equations. Second, set up the frequency-domain perturbation solver and choose the PARDISO solver; the PARDISO solver is better suited for solving the acoustic problem than the default MUMPS solver.

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

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

3 Clear the **Generate default plots** check box.

Step 1: Stationary

1 On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

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

3 In the table, enter the following settings:

Physics	Solve for	Discretization
Thermoacoustics, Frequency Domain	×	physics

Step 2: Frequency-Domain, Perturbation

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Frequency-Domain, Perturbation**.

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

3 In the **Frequencies** edit field, type `10^{range(2,2/5,4)} 10^{range(4.1,1.4/30,5.5)}`.

This gives a frequency range of 100 Hz-316 kHz. The reason for including such high frequencies is to be able to observe the fall-off in sensitivity.

Solver 1

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

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1>Stationary Solver 2** node, then click **Direct 1**.

3 In the **Direct** settings window, locate the **General** section.

4 From the **Solver** list, choose **PARDISO**.

5 Click the **Compute** button.

RESULTS

Derived Values

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

2 In the **Global Evaluation** settings window, locate the **Expression** section.

3 In the **Expression** edit field, type `es.V0_1`.

4 From the **Expression evaluated for** list, choose **Static solution**.

5 Click the **Evaluate** button.

The static polarization voltage across the membrane should equal 101.5 V. Tweaking the value of the membrane charge Qm changes this voltage. It is not possible to both directly set the static polarization voltage (equal to Vpol) and measure the output voltage of the microphone. An indirect way is to set up a global constraint on the terminal charge Qm.

RESULTS

2D Plot Group 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Model Builder** window, under **Results** right-click **2D Plot Group 1** and choose **Surface**.

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

4 Click the **Replace Expression** button in the upper-right corner of the section and select **Thermoacoustics, Frequency Domain>Instantaneous local velocity (ta.v_inst)**. Clear the **Compute differential** check box.

5 On the **2D plot group** toolbar, click **Plot**.

6 In the **Model Builder** window, right-click **2D Plot Group 1** and choose **Rename**.

7 Go to the **Rename 2D Plot Group** dialog box and type `2D Velocity` in the **New name** edit field.

8 Click **OK**.

To get a better view of the long slender geometry, disable the Preserve aspect ratio option.

DEFINITIONS

Axis

1 In the **Model Builder** window, expand the **View 1** node, then click **Axis**.

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

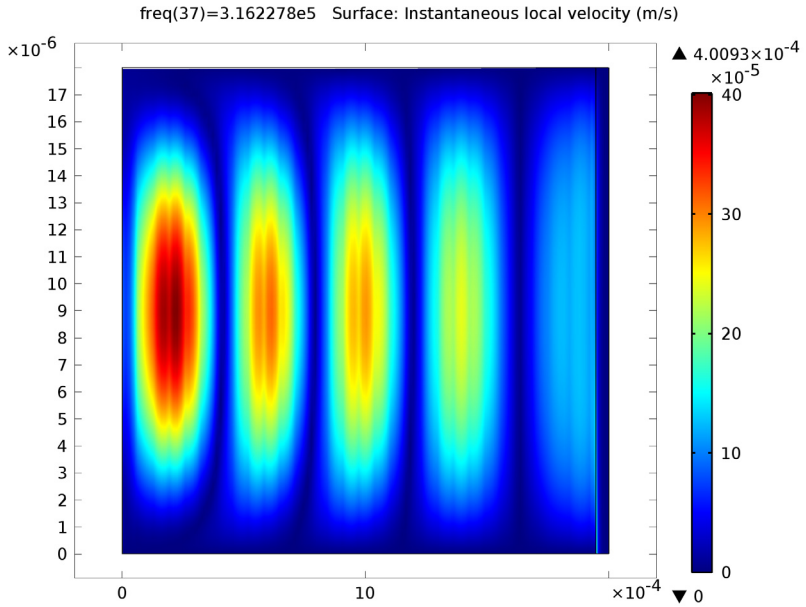
3 Clear the **Preserve aspect ratio** check box.

4 Click the **Apply** button.

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

RESULTS

2D Velocity



2D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

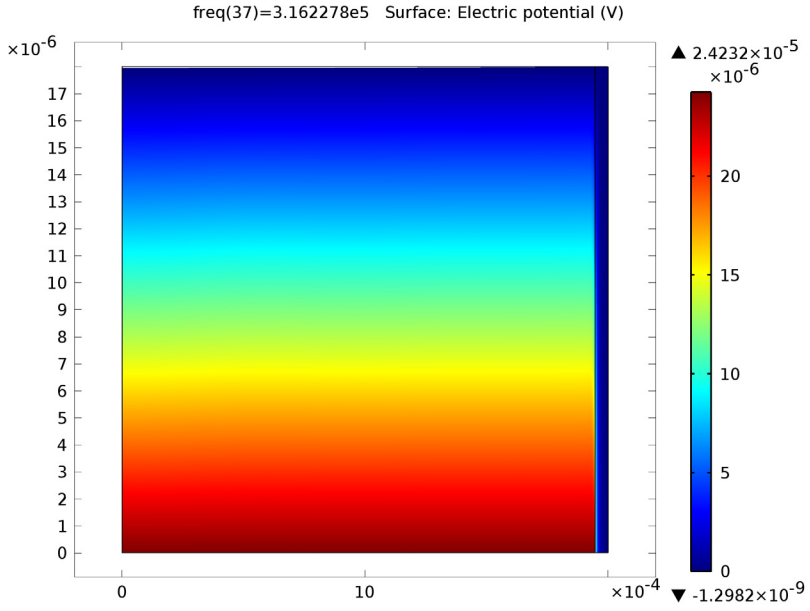
2 In the **Model Builder** window, under **Results** right-click **2D Plot Group 2** and choose **Surface**.

3 Right-click **2D Plot Group 2** and choose **Rename**.

4 Go to the **Rename 2D Plot Group** dialog box and type `2D Potential` in the **New name** edit field.

5 Click **OK**.

6 On the **2D plot group** toolbar, click **Plot**.



1D Plot Group 3

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

2 On the **1D plot group** toolbar, click **Global**.

3 In the **Global** settings window, locate the **y-Axis Data** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`20*log10(abs(es.V0_1/pin))`		`Sensitivity`

5 Clear the **Compute differential** check box.

6 Click the **x-Axis Log Scale** button on the Graphics toolbar.

7 In the **Model Builder** window, click **1D Plot Group 3**.

8 In the **1D Plot Group** settings window, click to expand the **Legend** section.

9 From the **Position** list, choose **Lower left**.

10 Locate the **Plot Settings** section. Select the **x-axis label** check box.

11 In the associated edit field, type `f (Hz)`.

12 Select the **y-axis label** check box.

13 In the associated edit field, type `Sensitivity (dB rel. 1 V/Pa)`.

14 On the **1D plot group** toolbar, click **Plot**.

15 Right-click **1D Plot Group 3** and choose **Rename**.

16 Go to the **Rename 1D Plot Group** dialog box and type `Sensitivity` in the **New name** edit field.

17 Click **OK**.

The microphone sensitivity curve should look like Figure 2.

Now, plot the membrane deformation, both the static and the harmonic components. Finally, plot the membrane deformation on a 3D revolved geometry using a 2D revolution data set.

1D Plot Group 4

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

2 In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Rename**.

3 Go to the **Rename 1D Plot Group** dialog box and type `Membrane deformation` in the **New name** edit field.

4 Click **OK**.

Membrane deformation

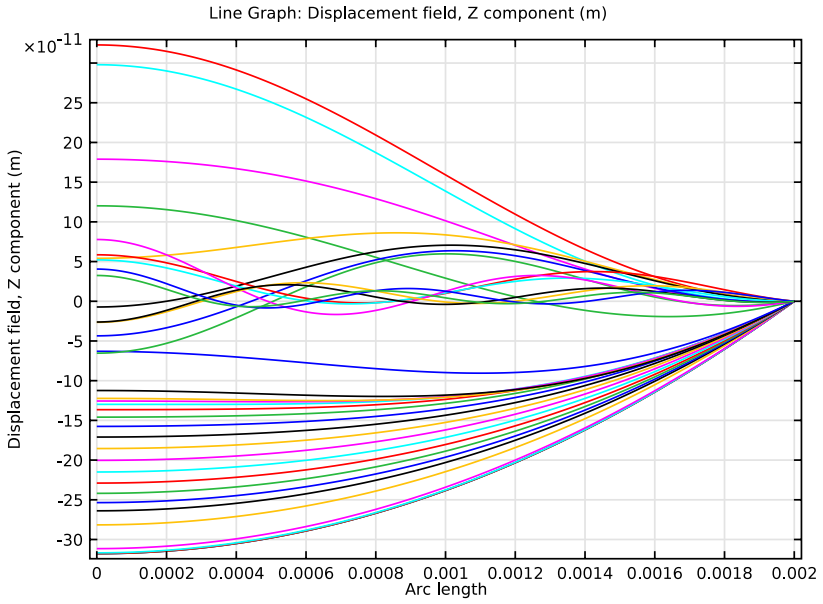
1 On the **1D plot group** toolbar, click **Line Graph**.

2 In the **Line Graph** settings window, locate the **Selection** section.

3 From the **Selection** list, choose **Membrane**.

4 Locate the **y-Axis Data** section. In the **Expression** edit field, type `wm`.

5 On the **1D plot group** toolbar, click **Plot**.



Membrane deformation 1

1 In the **Model Builder** window, right-click **Membrane deformation** and choose **Duplicate**.

2 Right-click **Membrane deformation 1** and choose **Rename**.

3 Go to the **Rename 1D Plot Group** dialog box and type `Static membrane deformation` in the **New name** edit field.

4 Click **OK**.

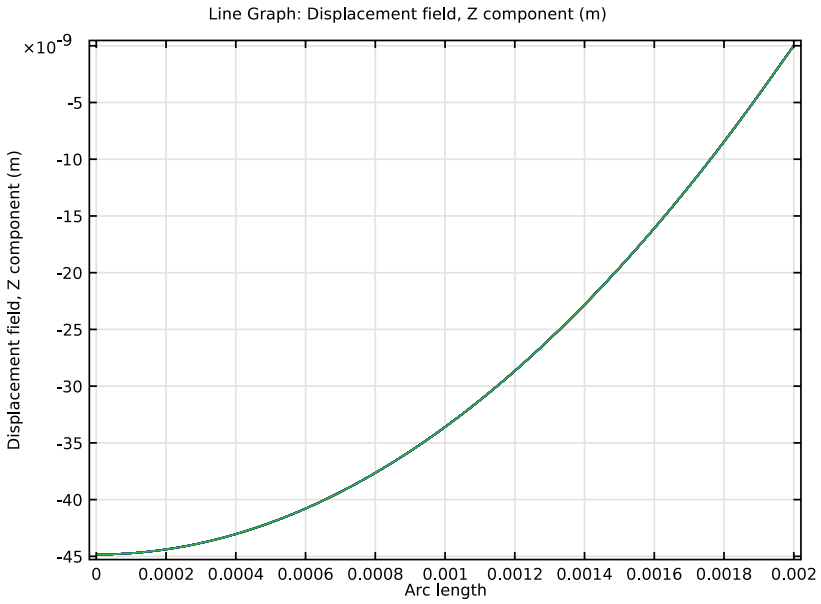
Static membrane deformation

1 In the **Model Builder** window, expand the **Results>Static membrane deformation** node, then click **Line Graph 1**.

2 In the **Line Graph** settings window, locate the **y-Axis Data** section.

3 From the **Expression evaluated for** list, choose **Static solution**.

4 On the **1D plot group** toolbar, click **Plot**.



Data Sets

On the **Results** toolbar, click **More Data Sets** and choose **Revolution 2D**.

3D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Model Builder** window, under **Results** right-click **3D Plot Group 6** and choose **Rename**.

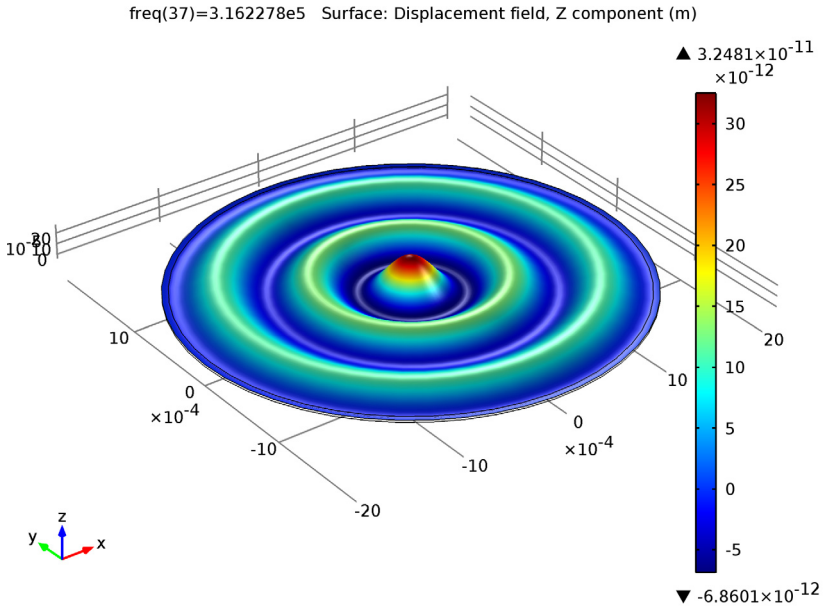
3 Go to the **Rename 3D Plot Group** dialog box and type `3D Membrane deformation` in the **New name** edit field.

4 Click **OK**.

3D Membrane deformation

1 Right-click **Results>3D Plot Group 6** and choose **Surface**.

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

3 Click **Displacement field, Z component (wm)** in the upper-right corner of the section. Right-click **Results>3D Membrane deformation>Surface 1** and choose **Deformation**.

4 On the **3D plot group** toolbar, click **Plot**.