Created in COMSOL Multiphysics 5.6

Axisymmetric Condenser Microphone with Electrical Lumping

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, using lumped descriptions for some of the physical phenomena. Lumping certain parts of a model can give additional insight to physical phenomena using a simplified description. A fully coupled approach (without lumping internal physics) can be seen in the Axisymmetric Condenser Microphone model, also located in the Application Library of the Acoustics Module.

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 electromechanical acoustic transducer works by transforming the mechanical deformation of a thin membrane (diaphragm) into an AC voltage signal.

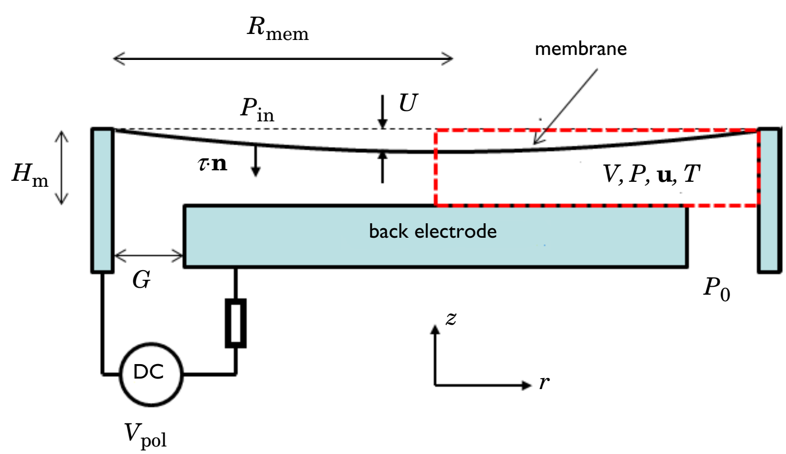


Figure 1: Sketch of the condenser microphone system including variables and coordinate system. The red box indicates the modeled region.

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. In the present detailed finite-element (FE) model, you model the microphone including a static (quiescent) analysis of the DC charging (prepolarization) and deformation of the membrane. You then perform time-harmonic (small-signal) finite element (FE) analysis of the dynamics of the membrane coupled to thermoviscous acoustics. The small-signal electric model for the system is solved as a lumped (electric equivalent) model coupled to the FE model. The model is a true multiphysics problem that involves several physics interfaces: Thermoviscous Acoustics, Electrostatics, Moving Mesh, two user-defined PDE interfaces, a Global ODEs and DAEs interface, as well as an Electrical Circuit model.

Note: This application requires the AC/DC Module.

Model Definition

The geometry and model definitions are shown in Figure 1. 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.

PARAMETER	VALUE	DESCRIPTION
H_{m}	18 µm	Air gap thickness
R_{mem}	2 mm	Membrane radius
G	54 µm	Slit gap width
T_{m}	3150 N/m	Membrane tension
t_{m}	7 µm	Membrane thickness
ρ_{m}	8300 kg/m^3	Membrane density
V_{pol}	100 V	Polarization voltage

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 thermoviscous acoustics interface. The membrane and back electrode makes up a capacitor that is polarized by an external DC voltage source. 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 and is coupled to the capacitor (the quiescent DC capacitance of the system C_0) and an external very large resistive load R_L.

The electric circuit for this coupling is shown in Figure 2.

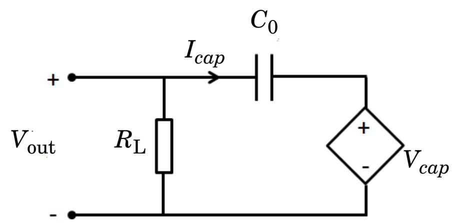


Figure 2: Analogous circuit for the electrical part of the condenser microphone.

The sensitivity L of the condenser microphone 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

$$L = 20\log\left(\frac{V_{out}}{p_{in}}\right)$$

Next, consider a small-signal (linearized) analysis of the electric part of the condenser microphone. The total voltage is the sum of the quiescent polarization voltage V_{pol} and the small-signal output voltage V_{out}. The total charge on the condenser is the sum of the quiescent charge Q and the small-signal charge q. The distance between the back electrode and the membrane is the sum of the initial distance H_m, the quiescent average deformation $U_{0,av}$ and the small-signal deformation U_{av}.

The capacitance C of a parallel plate capacitor, with a fixed air gap distance h and area A is given by

$$C = \frac{\varepsilon_0\varepsilon_r A}{h}$$

If the air gap h is varied with an average deformation U_{av} around the initial static gap distance, $H_m + U_{0,av}$, the expression becomes

$$C = \frac{\varepsilon_0\varepsilon_r A}{H_m + U_{0,av} + U_{av}} = \frac{\varepsilon_0\varepsilon_r A}{H_m + U_{0,av}}\left(1 - \frac{U_{av}}{H_m + U_{0,av}}\right) + O(U_{av}^2) \tag{1}$$

where the expression has been expanded to first order. Again, the distance $U_{0,av}$ stems from the initial deformation from equilibrium due to the electrostatic forces of the prepolarization of the condenser. A first approximation for the average deformation in axisymmetric coordinates is given as

$$U_{\mathrm{av}} = \frac{1}{A}\int U dA = \frac{2}{R_{\mathrm{mem}}^2}\int_0^{R_m - G} U(r) r dr \tag{2}$$

The capacitance is by its definition (see Ref. 4) given by

$$Q = CV \tag{3}$$

where Q is the charge on and V the potential across the capacitor. Inserting Equation 1 into Equation 3 and retaining only first-order terms yields

$$V_{\mathrm{pol}} + V_{\mathrm{out}}(t) = \frac{Q + q(t)}{C} \cong \frac{Q}{C_0}\left(1 + \frac{U_{\mathrm{av}}(t)}{H_{\mathrm{m}} + U_{0,\,\mathrm{av}}}\right) + \frac{q(t)}{C_0} \tag{4}$$

Differentiating Equation 4 with respect to time, switching to frequency domain, and using $V_{\mathrm{pol}} = Q/C_0$ yields

$$V_{\mathrm{out}} = \frac{I_{\mathrm{cap}}}{i\omega C_0} + \frac{u_{\mathrm{av}} V_{\mathrm{pol}}}{i\omega(H_{\mathrm{m}} + U_{0,\,\mathrm{av}})} \tag{5}$$

where I_{cap} is the induced current through the capacitor, the second term on the right is the electromechanical coupling V_{cap}, and the average membrane velocity is

$$u_{\mathrm{av}} = \frac{1}{A}\int u_{\mathrm{m}} dA = \frac{2}{R_{\mathrm{mem}}^2}\int_0^{R_m - G} u_{\mathrm{m}}(r) r dr \tag{6}$$

where u_{m} is the axial membrane velocity. The circuit model equivalent to equation Equation 5 is shown in Figure 2.The governing equation of the membrane is described in the next section, Membrane Model. The final element necessary to couple the lumped small parameter model of the electric model to the mechanical FE model is the back coupling via the small parameter electrostatic force f_{es}. The force is approximated by the spatial derivative (z direction) of the electric energy stored in a parallel plate condenser; the small parameter component is (see Ref. 2)

$$f_{\mathrm{es}} = -\frac{V_{\mathrm{pol}} I_{\mathrm{cap}}}{i\omega(H_{\mathrm{m}} + U_{0,\,\mathrm{av}})}$$

The force is applied evenly over the membrane as a surface normal stress $f_{\mathrm{es}}/(2\pi R_{\mathrm{mem}}^2)$.

Note that the integrals in Equation 2 and Equation 6 are over the area of the membrane that is backed by the back electrode plus a possible small correction for edge effects. This is especially important if the back electrode has holes. This is the case in many commercials condenser microphones where the holes are placed in order to produce a special sensitivity characteristic of the microphone. In this model the back electrode is flat and uniform.

MEMBRANE MODEL

The displacement U of a thin axisymmetric membrane of thickness t_m, under constant tension T_m, and with a density ρ_m is governed by the following equation

$$T_m \frac{\partial}{\partial r}\left(r\frac{\partial U}{\partial r}\right) - \rho_{ms} r \frac{\partial^2 U}{\partial t^2} - rF_s = 0 \tag{7}$$

where r is the radial coordinate, t is time, $\rho_{ms} = \rho_m/t_m$ is the surface density, and F_s is the sum of surface forces; see for example Ref. 3. In the present model, the surface force is the sum of the external incident pressure p_{in} (it is assumed to be uniform over the microphone membrane), the internal pressure $p = p(\mathbf{r})$ (given by the thermoviscous acoustics model), and the electrostatic force which is the sum of the quiescent Maxwell surface stress $\mathbf{n} \cdot \tau$ (given by the electrostatic model) and the small-signal force f_{es}. The variations of the deformation U is assumed to be small and harmonic on top of the static contribution U_0 from the DC polarization, such that

$$\begin{aligned} U(\mathbf{r}, t) &= U_0(\mathbf{r}) + U(\mathbf{r})e^{i\omega t} \\ p_{in}(\mathbf{r}, t) &= p_{in}e^{i\omega t} \\ p(\mathbf{r}, t) &= p(\mathbf{r})e^{i\omega t} \\ F_{es}(\mathbf{r}, t) &= \mathbf{n} \cdot \tau + f_{es}/(2\pi R_m^2) \cdot e^{i\omega t} \end{aligned}$$

Using these expressions, Equation 7 is reformulated into a static and a time-harmonic equation as

$$\begin{aligned} T_m \frac{\partial}{\partial r}\left(r\frac{\partial U_0}{\partial r}\right) - r(\mathbf{n} \cdot \tau) &= 0 \\ T_m \frac{\partial}{\partial r}\left(r\frac{\partial U}{\partial r}\right) + \rho_{ms} r\omega^2 U - r(f_{es}/(2\pi R_{mem}^2) + p_{in} - p) &= 0 \end{aligned} \tag{8}$$

The latter equation may be rewritten in terms of the axial velocity, $u_m = i\omega U$, of the membrane in the form of a Helmholtz equation:

$$T_m \frac{\partial}{\partial r}\left(r \frac{\partial u_m}{\partial r}\right) + T_m k_m^2 r u_m - i\omega r(p_{in} - p) = 0$$
$$k_m^2 = \frac{\omega^2 \rho_{ms}}{T_m} \tag{9}$$

Here k_m is the membrane wave number. In this model you disregard the change in tension due to the movement of the membrane, which is a nonlinear effect that is small compared to the tension T_m.

Results and Discussion

This model involves a detailed description of the physical effects at play in a simple condenser microphone. The lumping of the small-signal analysis of the electrical part is a good approximation for this simple geometry, where the back electrode is flat and has no perforations. The sensitivity, L, of the microphone is directly determined from the model (voltage across the load resistance divided by the incident pressure) and is shown in Figure 3.

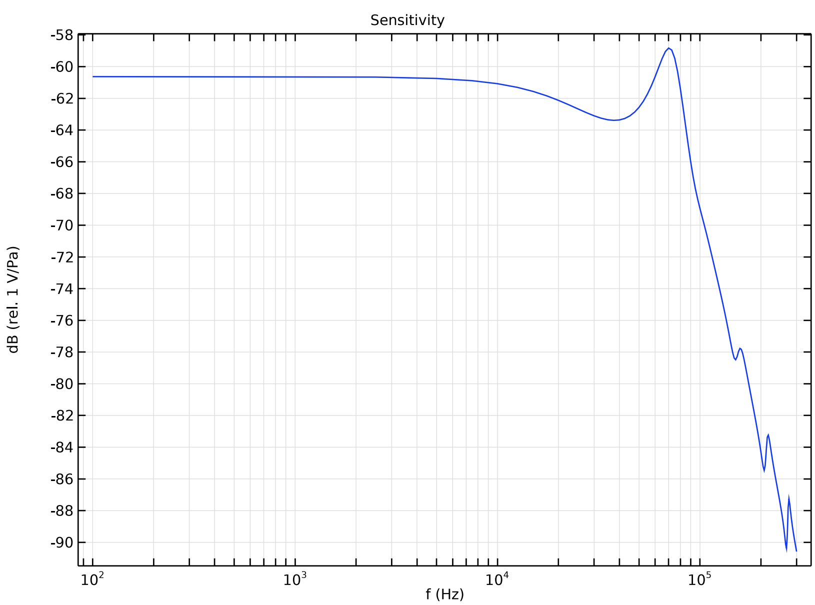


Figure 3: Sensitivity curve of the microphone measured in dB relative to 1 V/Pa.

For the case of the simple geometry used in this model, an analytical solution exists for the dynamics of the undamped membrane; see Ref. 5. The axial displacement is given by

$$U_{\mathrm{th}}(r) = \frac{p_{\mathrm{in}}}{T_{\mathrm{m}} k_{\mathrm{m}}^2}\left(1 - \frac{J_0(k_{\mathrm{m}} r)}{J_0(k_{\mathrm{m}} R_{\mathrm{mem}})}\right)$$

where k_{m} is the wave number defined in Equation 9. The analytical approximation is compared to the model results in Figure 4, which shows the average deformation versus frequency. The results agree well below the resonance frequency of the system. The average behavior above the first resonance (in between resonances) is also well captured by the approximate theoretical model. In the real system the damping introduced by the thermal and viscous losses in the air gap is important, especially at the resonances. This is also seen from the figure, where the resonance of the full (real) system is damped and shifted in frequency. The comparison of the two models is used as an extra indicator for the correctness of the FE model.

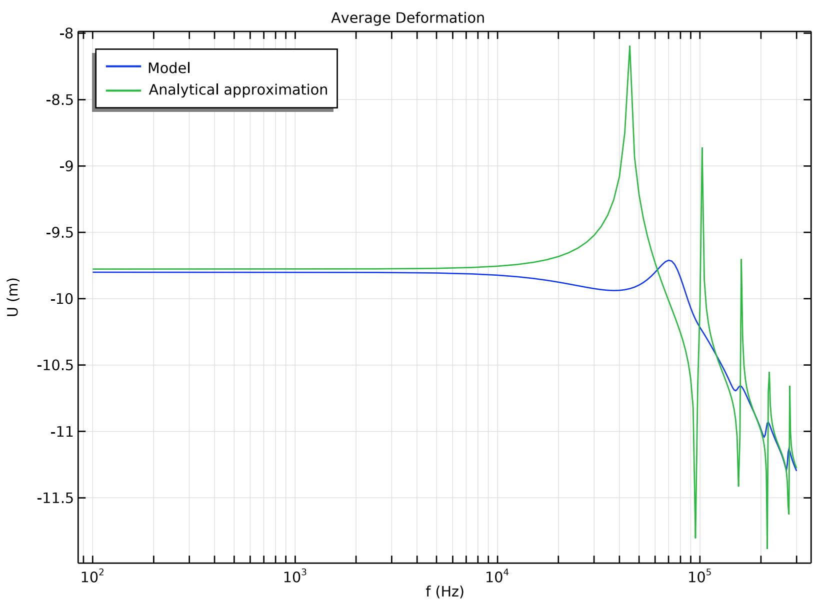


Figure 4: Comparison of the average membrane deformation given by the COMSOL model and by the theoretical approximation for the undamped membrane.

The shape of the deformed membrane is plotted for f = 0.3 GHz as a 3D surface in Figure 5 using a revolution 2D data set. At this frequency it is clear to see how higher order modes in the membrane are the cause of the poor sensitivity.

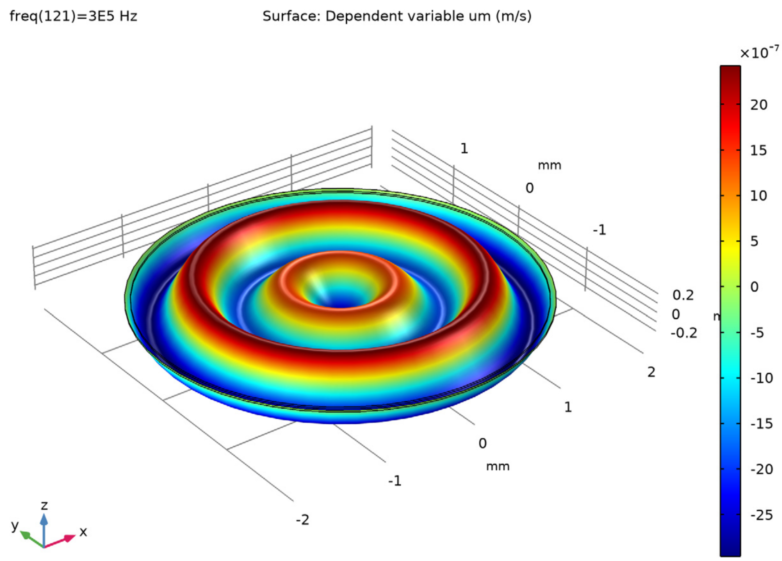


Figure 5: 3D representation of the harmonic membrane deformation at 0.3 GHz.

The principles described in this model may be extended to 3D models with more complex geometries where, for example, the back electrode is perforated or has a convex shape. Such a model may 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

The implementation couples a static model that determines the quiescent shape of the membrane after the polarization voltage is applied to the time harmonic small-signal response. The current model does not consider the transient charging of the condenser. The first step requires solving an electrostatic model (AC/DC interface) coupled to the static membrane model (user defined PDE model). In order for the model to determine the correct quiescent capacitance C_0, a Moving Mesh interface is necessary, as the capacitance is a geometric dependent quantity.

The second step is to solve the frequency domain model that describes the time harmonic small-signal deformation of the membrane (user defined PDE) and the interaction with the fluid (thermoviscous acoustics model) within the microphone. This solution is superposed to the static solution. The small-signal electric components of the microphone and the sensitivity is determined by a small lumped AC/DC circuit model.

STATIONARY SURFACE CHARGE AND CAPACITANCE

In the variables list the static capacitance variable C0 is defined as:

```
es.term1.int(es.nD*2*pi*r)/es.V0_1
```

This term is equal to the stationary terminal charge Q_0 divided bu the stationary voltage V_0. The stationary charge needs to be calculated as a surface integral of the D field and not evaluated using the variable es.Q0_1. This variable evaluates to 0 in the frequency domain step due to the current way reaction forces are transferred between study steps.

WEAK FORM OF THE MEMBRANE EQUATION

The membrane equations are implemented in COMSOL using the general weak form formulation of a partial differential equation. This is an integral form of the strong formulation of Equation 8. The equation is multiplied by a test function and integration by parts is performed (using Green's first identity). The resulting equation for the static deformation becomes

$$\int_{\partial\Omega} r\left(F_s\Phi - T_{\mathrm{m}}\frac{\partial U_0}{\partial r}\frac{\partial\Phi}{\partial r}\right)dl + \sum_N rT_{\mathrm{m}}\frac{\partial U_0}{\partial r}\Phi$$

where Φ is the test function of the displacement U_0. In COMSOL Multiphysics this is formulated as

```
r*((es.dnTez)*test(U0)-Tm*dtang(U0,r)*test(dtang(U0,r)))
```

The resulting weak form equation for the time-harmonic variation of the membrane reads

$$\int_{\partial\Omega} r\left((T_{\mathrm{m}}k_{\mathrm{m}}^2u_{\mathrm{m}} - i\omega(p_{\mathrm{in}}-p))\Phi - T_{\mathrm{m}}\frac{\partial u_{\mathrm{m}}}{\partial r}\frac{\partial\Phi}{\partial r}\right)dl + \sum_N rT_{\mathrm{m}}\frac{\partial u_{\mathrm{m}}}{\partial r}\Phi$$

which is formulated as

```
r*((Tm*kmsq*um-ta.iomega*(pin-p))*test(um)-
    Tm*dtang(um,r)*test(dtang(um,r)))
```

The sum over N in both expressions are the boundary (point) contribution at the center $r = 0$ and the edge $r = R_{\mathrm{mem}}$ of the membrane.

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. D.J. Griffiths, *Introduction to Electrodynamics*, 3rd ed., Pearson Education, 2008.

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

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/condenser_microphone_lumped`

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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Acoustics>Thermoviscous Acoustics>Thermoviscous Acoustics, Frequency Domain (ta)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **AC/DC>Electric Fields and Currents>Electrostatics (es)**.

5 Click **Add**.

6 In the **Select Physics** tree, select **AC/DC>Electrical Circuit (cir)**.

7 Click **Add**.

8 In the **Select Physics** tree, select **Mathematics>PDE Interfaces>Lower Dimensions>Weak Form Boundary PDE (wb)**.

9 Click **Add** twice.

10 Click **Study**.

11 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Electrostatics>Small-Signal Analysis, Frequency Domain**.

12 Click **Done**.

GLOBAL DEFINITIONS

A set of parameters defining the material properties and the geometry are available in a text file that can be loaded.

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `condenser_microphone_lumped_parameters.txt`.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `Rmem`.

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

5 Click **Build Selected**.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

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

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

5 Locate the **Position** section. In the **r** text field, type `Rmem-G`.

6 Click **Build Selected**.

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

DEFINITIONS

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `condenser_microphone_lumped_variables.txt`.

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_be` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 3 only.

5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

Membrane

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

2 In the **Settings** window for **Explicit**, type `Membrane` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 3 and 6 only.

Deforming Domain 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Deforming Domain**.

2 In the **Settings** window for **Deforming Domain**, locate the **Domain Selection** section.

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

Fixed Boundary 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Fixed Boundary**.

2 Select Boundaries 2, 5, and 7 only.

Prescribed Mesh Displacement 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Prescribed Mesh Displacement**.

2 Select Boundaries 3 and 6 only.

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

4 Specify the *dx* vector as

0	R
U0	Z

Symmetry/Roller 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Symmetry/Roller**.

2 Select Boundary 1 only.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Thermoviscous Acoustics Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Thermoviscous Acoustics, Frequency Domain (ta)** click **Thermoviscous Acoustics Model 1**.

2 In the **Settings** window for **Thermoviscous Acoustics Model**, locate the **Fluid Properties** section.

3 From the $\rho_0(p_{0,T0})$ list, choose **Ideal gas**.

4 From the **Gas constant type** list, choose **Mean molar mass**.

Pressure (Adiabatic) 1

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

2 Select Boundary 5 only.

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

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

Velocity 1

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

2 In the **Settings** window for **Velocity**, 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_{0z} text field, type `um`.

7 Click the **Show More Options** button in the **Model Builder** toolbar.

8 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

9 Click **OK**.

10 In the **Settings** window for **Velocity**, click to expand the **Constraint Settings** section.

11 From the **Apply reaction terms on** list, choose **Individual dependent variables**.

This setting is necessary because an influence of the acoustics on the membrane (the reaction force) is introduced via the pressure term, p, in the equation for the membrane that you will set up shortly. If this option were not selected, the pressure (or a scaled version thereof) would act twice on the membrane.

The thermal boundary condition for the membrane should be isothermal. This conditions is automatically selected in the wall condition for the walls.

Isothermal 1

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

2 In the **Settings** window for **Isothermal**, locate the **Boundary Selection** section.

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

ELECTROSTATICS (ES)

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

Terminal 1

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

2 Select Boundary 2 only.

3 In the **Settings** window for **Terminal**, locate the **Terminal** section.

4 From the **Terminal type** list, choose **Voltage**.

5 In the V_0 text field, type `Vpol`.

Ground 1

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

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

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

You have now set up the electrical circuit representing the small-signal part of the electrical model in accordance with the circuit depicted in Figure 2.

ELECTRICAL CIRCUIT (CIR)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electrical Circuit (cir)**.

Capacitor C1

1 In the **Electrical Circuit** toolbar, click **Capacitor**.
2 In the **Settings** window for **Capacitor**, locate the **Node Connections** section.
3 In the table, enter the following settings:

Label	Node names
p	1
n	2

4 Locate the **Device Parameters** section. In the C text field, type `C0`.

Resistor R1

1 In the **Electrical Circuit** toolbar, click **Resistor**.
2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.
3 In the table, enter the following settings:

Label	Node names
p	2
n	0

4 Locate the **Device Parameters** section. In the R text field, type `RL`.

Voltage Source V1

1 In the **Electrical Circuit** toolbar, click **Voltage Source**.
2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.
3 In the table, enter the following settings:

Label	Node names
p	1
n	0

4 Locate the **Device Parameters** section. From the **Source type** list, choose **AC-source**.
5 In the V_{src} text field, type `Vcap`.

WEAK FORM BOUNDARY PDE (WB)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Weak Form Boundary PDE (wb)**.

2 In the **Settings** window for **Weak Form Boundary PDE**, locate the **Boundary Selection** section.

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

4 Locate the **Units** section. In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	1

5 Click to expand the **Dependent Variables** section. In the **Field name** text field, type `U0`.

6 In the **Dependent variables** table, enter the following settings:

U0

7 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

8 In the **Physical Quantity** dialog box, type `displacement` in the text field.

9 Click **Filter**.

10 In the tree, select **General>Displacement (m)**.

11 Click **OK**.

Weak Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Weak Form Boundary PDE (wb)** click **Weak Form PDE 1**.

2 In the **Settings** window for **Weak Form PDE**, locate the **Weak Expressions** section.

3 In the weak text field, type `r*((es.dnTez)*test(U0)-Tm*dtang(U0,r)*test(dtang(U0,r)))`.

You can ignore the unexpected unit warning.

Constraint 1

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

2 Select Point 6 only.

3 In the **Settings** window for **Constraint**, locate the **Constraint** section.

4 In the R text field, type `-U0`.

WEAK FORM BOUNDARY PDE 2 (WB2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Weak Form Boundary PDE 2 (wb2)**.

2 In the **Settings** window for **Weak Form Boundary PDE**, locate the **Boundary Selection** section.

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

4 Locate the **Units** section. In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	1

5 Locate the **Dependent Variables** section. In the **Field name** text field, type `um`.

6 In the **Dependent variables** table, enter the following settings:

um

7 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

8 In the **Physical Quantity** dialog box, type `velocity` in the text field.

9 Click **Filter**.

10 In the tree, select **General>Velocity (m/s)**.

11 Click **OK**.

Weak Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Weak Form Boundary PDE 2 (wb2)** click **Weak Form PDE 1**.

2 In the **Settings** window for **Weak Form PDE**, locate the **Weak Expressions** section.

3 In the weak text field, type `r*((Tm*kmsq*um-ta.iomega*(Fes+pin-p))*test(um)-Tm*dtang(um,r)*test(dtang(um,r)))`.

Ignore the inconsistent unit warning.

Constraint 1

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

2 Select Point 6 only.

3 In the **Settings** window for **Constraint**, locate the **Constraint** section.

4 In the R text field, type `-um`.

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>Air**.

4 Click **Add to Component** in the window toolbar.

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

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

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

4 Click in the **Graphics** window and then press Ctrl+A to select both domains.

5 Click to expand the **Reduce Element Skewness** section. Select the **Adjust edge mesh** check box.

Distribution 1

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

2 Select Boundary 3 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

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

Distribution 2

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

2 Select Boundary 4 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

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

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

6 In the **Element ratio** text field, type 2.

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

Distribution 3

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

2 Select Boundary 6 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

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

Boundary Layers 1

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

2 In the **Settings** window for **Boundary Layers**, click to expand the **Transition** section.

3 Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 Select Boundary 4 only.

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

4 In the **Number of boundary layers** text field, type `5`.

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

6 In the **Thickness** text field, type `2[um]`.

7 Click **Build Selected**.

The mesh is built such that it resolves the acoustic boundary layer at the maximal frequency of 320 kHz. At this frequency the viscous boundary layer is about 4 μm thick, corresponding to roughly 1/5 of the air-gap thickness.

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 In the table, clear the **Solve for** check boxes for **Thermoviscous Acoustics, Frequency Domain (ta)**, **Electrical Circuit (cir)**, and **Weak Form Boundary PDE 2 (wb2)**.

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 **Study Settings** section.

3 In the **Frequencies** text field, type `{100 range(2500,2500,300000)}`.

This gives a frequency range of 100 Hz - 300 kHz. The reason for including such high frequencies is to be able to observe the fall-off in sensitivity.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check boxes for **Electrostatics (es)** and **Weak Form Boundary PDE (wb)**.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Stationary Solver 2**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 From the **Linearity** list, choose **Automatic**.

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

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

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

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

RESULTS

Potential

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

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

Surface 1

1 Right-click **Potential** and choose **Surface**.

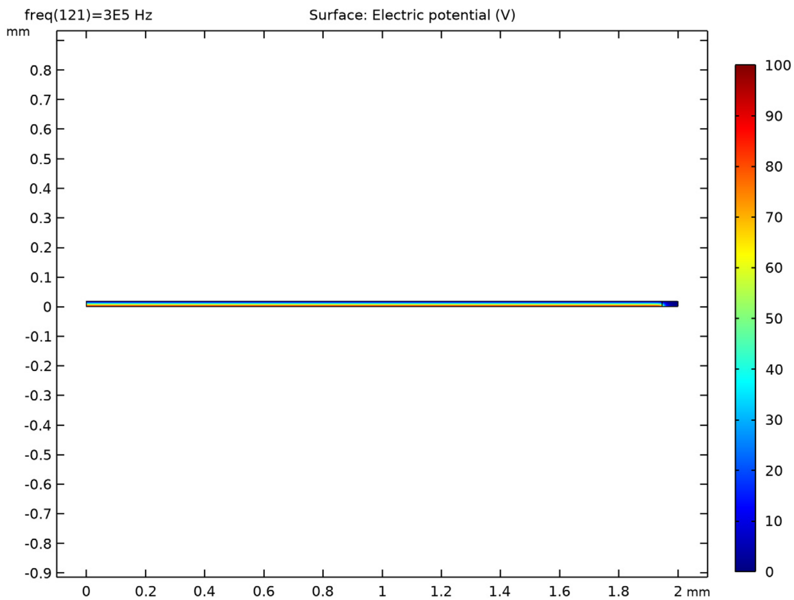
2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `V`.

4 In the **Potential** toolbar, click **Plot**.

Potential

You can examine the plot in greater detail by zooming around the edge of the model using the **Zoom Box** tool.



Velocity

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

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

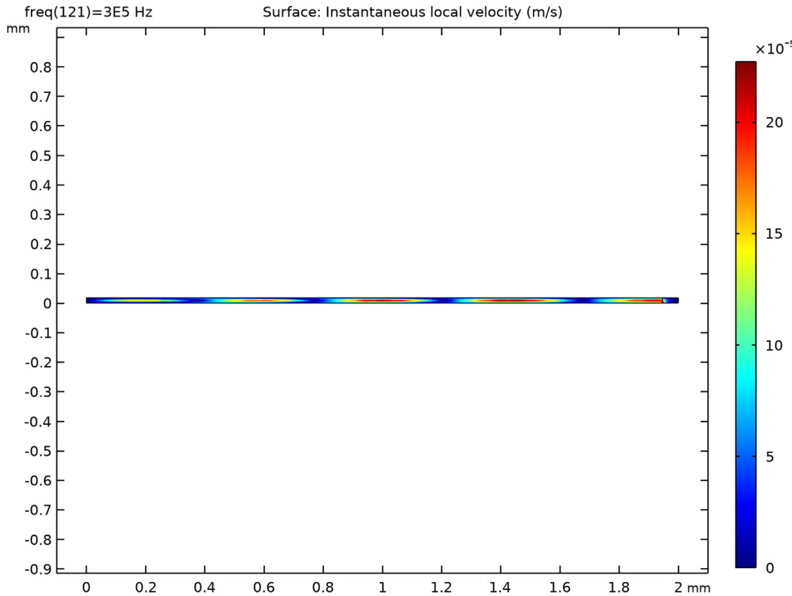
Surface 1

1 Right-click **Velocity** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Thermoviscous Acoustics, Frequency Domain>Acceleration and velocity>ta.v_inst - Instantaneous local velocity - m/s**.

3 In the **Velocity** toolbar, click  **Plot**.

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



Membrane Deformation

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

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

Line Graph 1

1 Right-click **Membrane Deformation** and choose **Line Graph**.

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

3 In the **Settings** window for **Line Graph**, locate the **Selection** section.

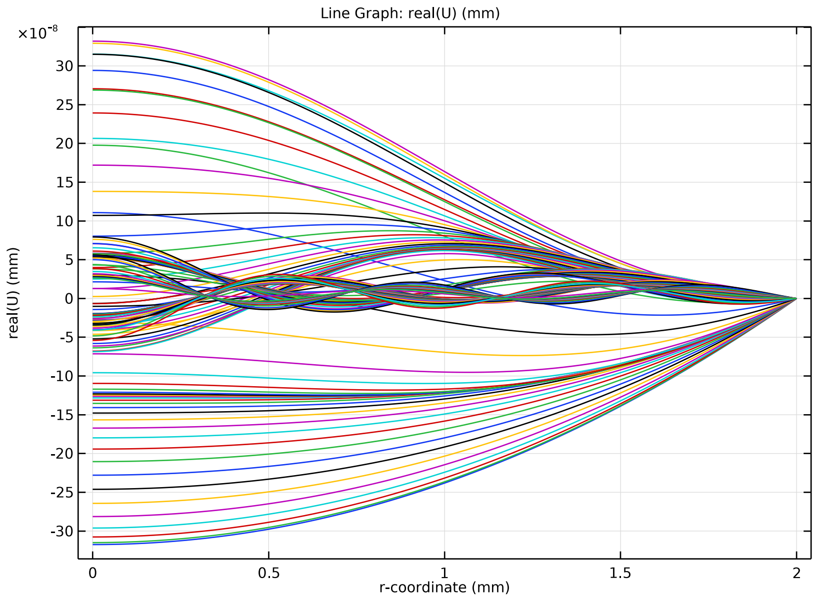
4 From the **Selection** list, choose **Membrane**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `real(U)`.

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

7 In the **Expression** text field, type `r`.

8 In the **Membrane Deformation** toolbar, click **Plot**.



Pressure Under Membrane

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

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

Line Graph 1

1 Right-click **Pressure Under Membrane** and choose **Line Graph**.

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

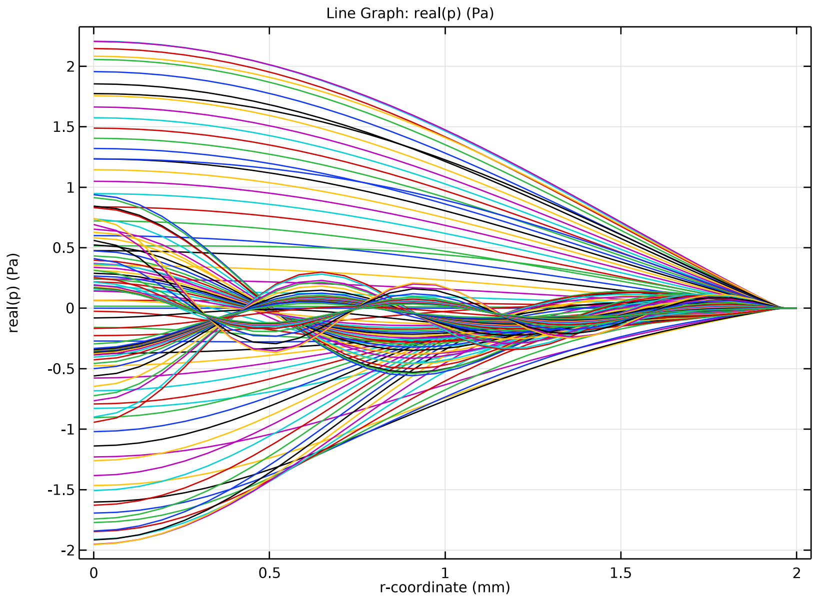
3 From the **Selection** list, choose **Membrane**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `real(p)`.

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

6 In the **Expression** text field, type `r`.

7 In the **Pressure Under Membrane** toolbar, click **Plot**.



Average Membrane Velocity

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

2 In the **Settings** window for **1D Plot Group**, type `Average Membrane Velocity` 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 `Average Membrane Velocity`.

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

6 In the associated text field, type `f (Hz)`.

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

8 In the associated text field, type `um_av (m/s)`.

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

Global 1

1 Right-click **Average Membrane Velocity** and choose **Global**.

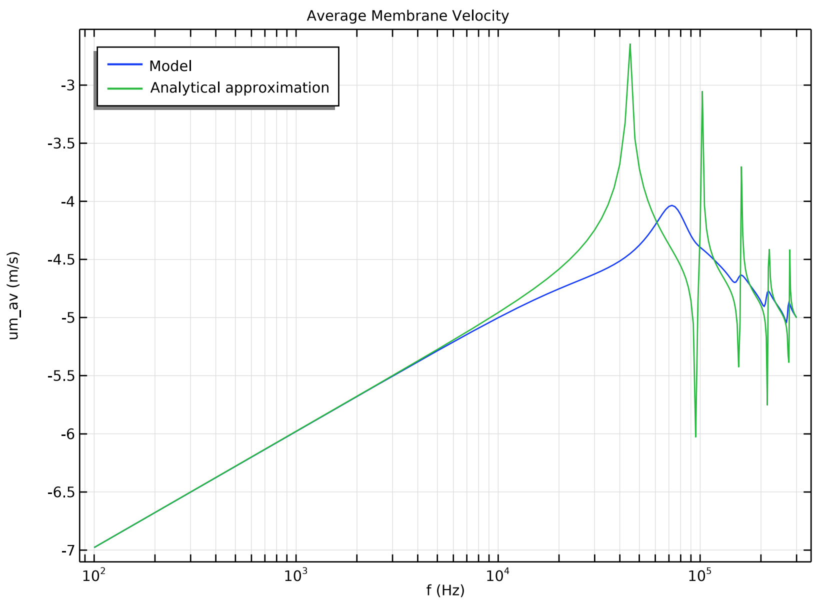
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`log10(abs(um_av))`		`Model`
`log10(abs(uth_av))`		`Analytical approximation`

4 In the **Average Membrane Velocity** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.



Average Membrane Deformation

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

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

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Average Deformation`.

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

6 In the associated text field, type `f (Hz)`.

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

8 In the associated text field, type `U (m)`.

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

Global 1

1 Right-click **Average Membrane Deformation** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`log10(abs(U_av))`		`Model`
`log10(abs(Uth_av))`		`Analytical approximation`

4 In the **Average Membrane Deformation** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

This reproduces the average membrane velocity plot depicted in Figure 4.

Sensitivity

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

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

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Sensitivity`.

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

6 In the associated text field, type `f (Hz)`.

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

8 In the associated text field, type `dB (rel. 1 V/Pa)`.

Global 1

1 Right-click **Sensitivity** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`20*log10(abs(cir.R1_v/pin))`		`Sensitivity`

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

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

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

This should reproduce the sensitivity curve depicted in Figure 3.

Finally, create a 2D revolution dataset to plot the membrane deformation on the revolved 3D geometry.

Revolution 2D 1

In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

3D Membrane Deformation

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `3D Membrane Deformation` in the **Label** text field.

Surface 1

1 Right-click **3D Membrane Deformation** and choose **Surface**.

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

3 In the **Expression** text field, type `um`.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.

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

3 In the **r component** text field, type `0`.

4 In the **phi component** text field, type `0`.

5 In the **z component** text field, type `um`.

6 In the **3D Membrane Deformation** toolbar, click **Plot**.

The plot should reproduce Figure 5.