Created in COMSOL Multiphysics 6.0

Solidly Mounted Resonator 2D

Introduction

A solidly mounted resonator (SMR) is a micromachined, piezoelectric MEMS resonator formed on top of an acoustic mirror on a thick substrate. The acoustic mirror comprises alternating layers of materials with high and low acoustic impedances, which confines acoustic energy within the piezoelectric active layer. SMRs are very robust and have higher resonant frequencies than Bulk Acoustic Wave (BAW) resonators (in the GHz range). This tutorial shows how you can model SMR in 2D using eigenfrequency and frequency-response analyses.

Model Definition

The geometry of the SMR model and its key components are shown in Figure 1.

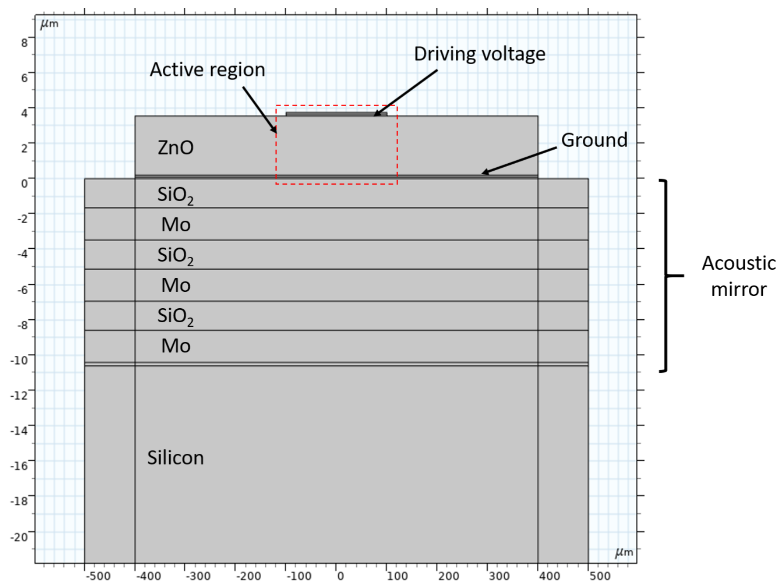


Figure 1: Model geometry showing the key components of the solidly mounted resonator.

Note that for clarity, the vertical scale is magnified to show the layers. All dimensions are parameterized in the model. Various selection features are used for the construction of the geometry and the setup of physics and mesh.

The fabrication of the device is discussed in Ref. 1. Here we provide a description of the final structure and an explanation of its principle of operation.

At the top of the device is a ZnO piezoelectric layer with aluminum electrodes at its top (drive) and bottom (ground) surfaces. Here, the pole direction is along the vertical axis and the piezoelectric material data is available in the built-in MEMS material library.

Underneath the piezoelectric resonator is a stack of alternating layers of molybdenum (high impedance) and silicon dioxide (low impedance). The thickness of the molybdenum and silicon dioxide layers were chosen to be 1.82 μm and 1.65 μm, respectively, to reflect the acoustic wave generated by the piezoelectric resonator and to prevent its dissipation in the silicon substrate. With this structure, the resonant frequency of the device is 870 MHz.

The parameters of the geometry are summarized in the first table in the section Modeling Instructions. The second table in the same section summarizes the material properties used in the model as specified in Ref. 1. Other material properties used in the model are obtained from the MEMS Module material library. As shown in the table, the Young's moduli of the materials and the wavelength in silicon are computed from the values of density and acoustic velocity listed in the paper.

In this model, the fully coupled structural and electrostatic equations are solved in the piezoelectric layer, while only the structural equation is solved in other layers. Electrostatics equations are not solved in the aluminum layers because of its high electrical conductivity.

Perfectly Matching Layers (PML) boundary conditions are used at the sides and the bottom of the device to introduce anchor damping and eliminate reflections. The model also includes mechanical losses through an isotropic structural loss factor of 1.5×10^{-4}. The model has fixed boundary conditions at the outer edges of the PML.

Note that although the structure has a plane of symmetry, which vertically bisects the device, we do not use a symmetry boundary condition because it would eliminate the anti-symmetric modes from the analysis.

This tutorial shows you the setup of Eigenfrequency and Frequency Domain studies. In the Eigenfrequency study you compute and investigate the eigenmodes of the structure. In the Frequency Domain study, 1 V drive signal is applied to the drive electrode and the frequency response of the resonator from 500 to 1200 MHz is analyzed.

To save time and reduce file size, a relatively coarse mesh is used, in particular in the horizontal direction. Thus only the main lower modes will be resolved in this model. The same approach was taken in the reference paper.

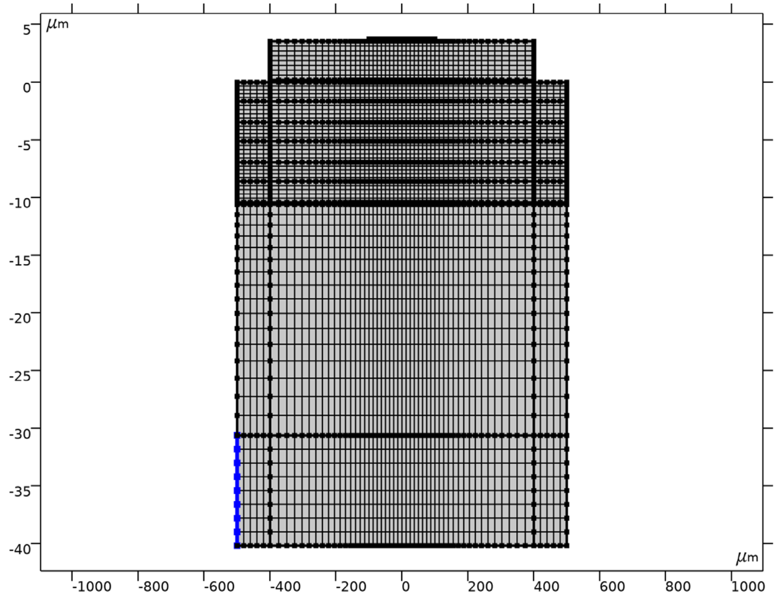


Figure 2: The mesh used in the model.

Results and Discussion

Figure 3 shows the mode shape of the fundamental mode of the resonator with the resonant frequency of about 870 MHz as intended by the design described in Ref. 1.

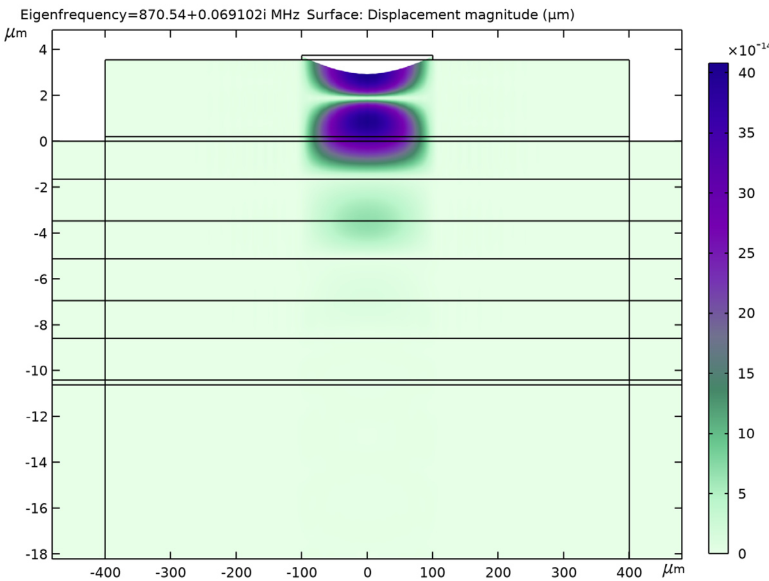


Figure 3: Surface plot of displacement indicating mode shape of the resonator's fundamental mode.

Figure 4 is a surface plot of the displacement showing the excitation of the fundamental mode of the resonator when driven at the resonant frequency of about 870 MHz. The plot agrees reasonably well with Fig. 5(a) in Ref. 1.

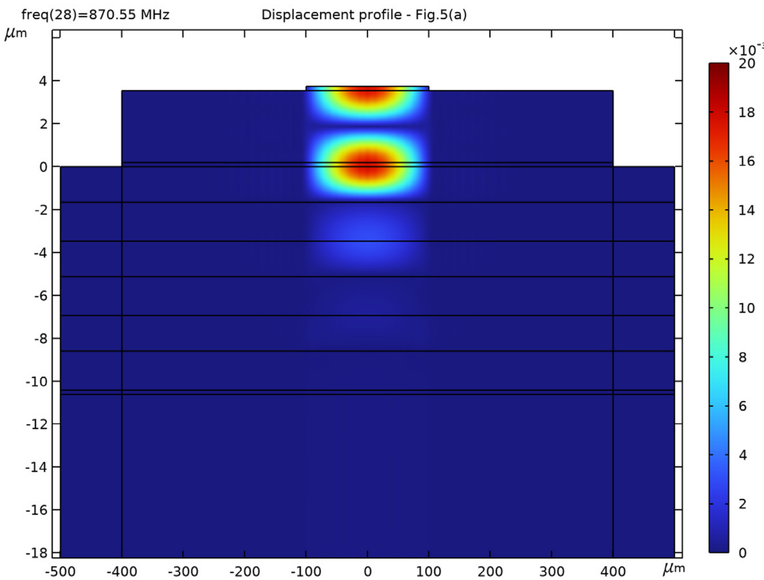


Figure 4: Surface plot of displacement indicating the excitation of the resonator’s fundamental mode.

Figure 5 is a surface plot of the electric potential corresponding to Figure 4.

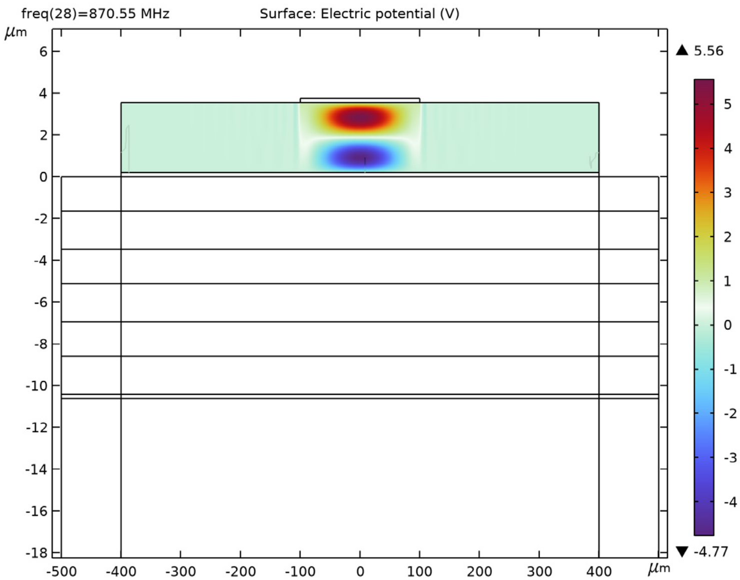


Figure 5: Surface plot of electric potential indicating the excitation of the resonator's fundamental mode.

Figure 6 is a line plot of the displacement as a function of depth showing the confinement of acoustic energy within the piezoelectric layer, in agreement with Figure 5(b) in Ref. 1.

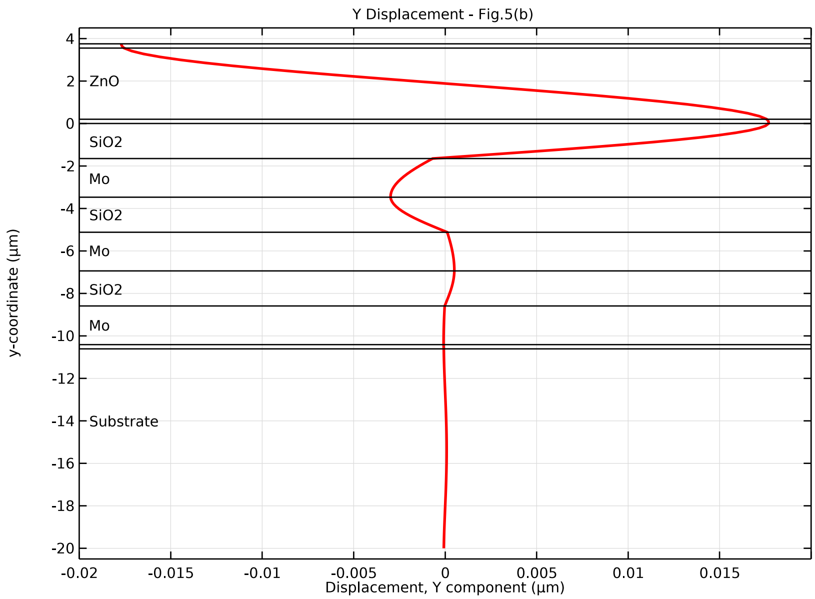


Figure 6: Displacement as function of depth showing the confinement of acoustic energy within the piezoelectric layer.

Figure 7 is a log plot of the magnitude of the impedance vs. frequency from 500 to 1200 MHz, in accordance with Figure 4 in Ref. 1.

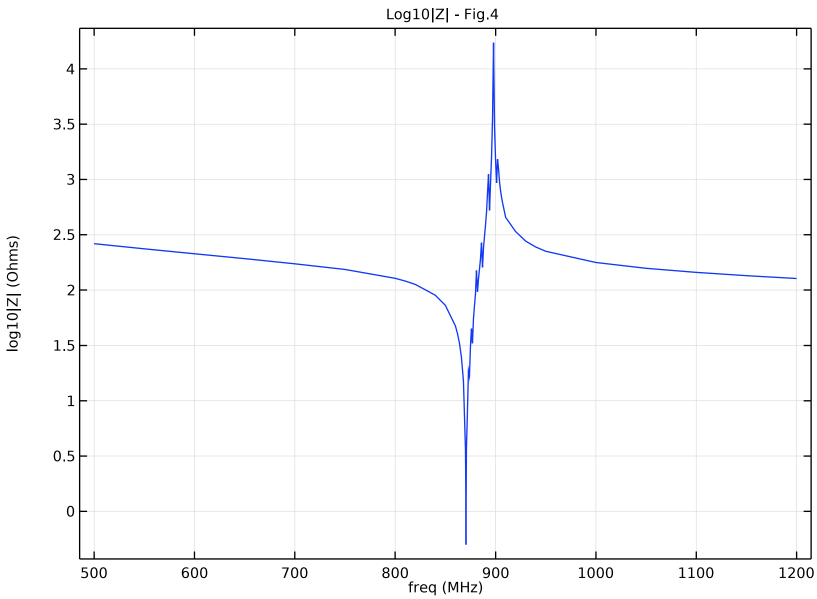


Figure 7: |Z| versus frequency from 500 to 1200 MHz.

Reference

1. F.H. Villa-López and others, "Design and Modelling of Solidly Mounted Resonators for Low-Cost Particle Sensing," *Measurement Science and Technology*, vol. 27, no. 2, 2016.

Application Library path: `MEMS_Module/Piezoelectric_Devices/solidly_mounted_resonator_2d`

Modeling Instructions

Start with a new 2D model with the built-in piezoelectric physics.

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

NEW

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

MODEL WIZARD

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

2 In the **Select Physics** tree, select **Structural Mechanics>Electromagnetics-Structure Interaction>Piezoelectricity>Piezoelectricity, Solid**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Eigenfrequency**.

6 Click **Done**.

GEOMETRY 1

Set the geometry unit to microns for convenience.

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 **µm**.

Since the device is much wider than its thickness, scale the graphics aspect ratio to zoom in in the thickness (Y) direction.

DEFINITIONS

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

Axis

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

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

3 From the **View scale** list, choose **Manual**.

4 In the **y scale** text field, type 35.

5 Click **Update**.

Enter geometry parameters. Note that we will truncate most of the thickness of the Si substrate and replace it with a perfectly matched layer (PML).

GLOBAL DEFINITIONS

Parameters 1 - Geometry

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

2 In the **Settings** window for **Parameters**, type `Parameters 1 - Geometry` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
t_s	500[um]/25	2E-5 m	Substrate thickness (truncated)
t_i	200[nm]	2E-7 m	Insulator thickness
t_hil	1.82[um]	1.82E-6 m	High impedance layer thickness
t_lil	1.65[um]	1.65E-6 m	Low impedance layer thickness
t_pe	3.35[um]	3.35E-6 m	Piezoelectric layer thickness
t_e	200[nm]	2E-7 m	Electrode thickness
w_ar	200[um]	2E-4 m	Active area width
w_pe	800[um]	8E-4 m	Piezoelectric layer width
w	1000[um]	0.001 m	Device width

Enter material parameters. Then calculate the Young's Modulus from the density and acoustic velocity for each linear material. Also calculate the wavelength in the substrate for an estimate of the PML thickness. A guessed value of 1.5e-4 is used for an isotropic damping factor to roughly match the vibration amplitude shown in the reference paper.

Parameters 2 - Material properties

1 In the **Home** toolbar, click Pi **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Parameters 2 - Material properties` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
rho_ZnO	5680[kg/m^3]	5680 kg/m³	Density of ZnO
rho_Mo	10200[kg/m^3]	10200 kg/m³	Density of Mo
rho_SiO2	2170[kg/m^3]	2170 kg/m³	Density of SiO2
rho_Al	2700[kg/m^3]	2700 kg/m³	Density of Al
rho_Si	2330[kg/m^3]	2330 kg/m³	Density of Si

Name	Expression	Value	Description
v_ZnO	6330[m/s]	6330 m/s	Acoustic velocity of ZnO
v_Mo	6280[m/s]	6280 m/s	Acoustic velocity_of Mo
v_SiO2	5540[m/s]	5540 m/s	Acoustic velocity of SiO2
v_Al	6450[m/s]	6450 m/s	Acoustic velocity of Al
v_Si	8320[m/s]	8320 m/s	Acoustic velocity of Si
E_Mo	rho_Mo*(v_Mo)^2	4.0227E11 Pa	Young's modulus of Mo
E_SiO2	rho_SiO2*(v_SiO2)^2	6.6601E10 Pa	Young's modulus of SiO2
E_Al	rho_Al*(v_Al)^2	1.1233E11 Pa	Young's modulus of Al
E_Si	rho_Si*(v_Si)^2	1.6129E11 Pa	Young's modulus of Si
eta0	1.5e-4	1.5E-4	Loss factor (guessed)
lambda_Si	v_Si/870[MHz]	9.5632E-6 m	Wavelength in Si

Build the parameterized geometry. Note how the selection and cumulative selection functionalities will be used to created named selections for material and physics settings later.

GEOMETRY 1

Piezo - ZnO

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

2 In the **Settings** window for **Rectangle**, type Piezo - ZnO in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type w_pe.

4 In the **Height** text field, type t_pe.

5 Locate the **Position** section. In the **x** text field, type -w_pe/2.

6 In the **y** text field, type t_e.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Top electrode - Al

1 Right-click **Piezo - ZnO** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, type `Top electrode - Al` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w_ar`.

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

5 Locate the **Position** section. In the **x** text field, type `-w_ar/2`.

6 In the **y** text field, type `t_pe+t_e`.

7 Locate the **Selections of Resulting Entities** section. From the **Show in physics** list, choose **All levels**.

8 Find the **Cumulative selection** subsection. Click **New**.

9 In the **New Cumulative Selection** dialog box, type `Al` in the **Name** text field.

10 Click **OK**.

Bottom electrode - Al

1 Right-click **Top electrode - Al** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, type `Bottom electrode - Al` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w_pe`.

4 Locate the **Position** section. In the **x** text field, type `-w_pe/2`.

5 In the **y** text field, type `0`.

Low impedance - SiO2

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

2 In the **Settings** window for **Rectangle**, type `Low impedance - SiO2` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w`.

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

5 Locate the **Position** section. In the **x** text field, type `-w/2`.

6 In the **y** text field, type `-t_lil`.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	**Thickness (µm)**
Layer 1	(w-w_pe)/2

8 Select the **Layers to the left** check box.

9 Select the **Layers to the right** check box.

10 Clear the **Layers on bottom** check box.

11 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

12 In the **New Cumulative Selection** dialog box, type `SiO2` in the **Name** text field.

13 Click **OK**.

Array - SiO2

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 In the **Settings** window for **Array**, type `Array - SiO2` in the **Label** text field.

3 Locate the **Input** section. From the **Input objects** list, choose **SiO2**.

4 Locate the **Size** section. In the **y size** text field, type `3`.

5 Locate the **Displacement** section. In the **y** text field, type `-t_lil-t_hil`.

High impedance - Mo

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Low impedance - SiO2 (r4)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, type `High impedance - Mo` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Height** text field, type `t_hil`.

4 Locate the **Position** section. In the **y** text field, type `-t_lil-t_hil`.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

6 In the **New Cumulative Selection** dialog box, type `Mo` in the **Name** text field.

7 Click **OK**.

Array - Mo

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Array - SiO2 (arr1)** and choose **Duplicate**.

2 In the **Settings** window for **Array**, type `Array - Mo` in the **Label** text field.

3 Locate the **Input** section. Find the **Input objects** subsection. Click to select the **Activate Selection** toggle button.

4 From the **Input objects** list, choose **Mo**.

5 Click **Build Selected**.

Insulator - SiO2

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Low impedance - SiO2 (r4)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, type `Insulator - SiO2` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Height** text field, type `t_i`.

4 Locate the **Position** section. In the **y** text field, type `-(t_lil*3)-(t_hil*3)-t_i`.

Substrate - Si

1 Right-click **Insulator - SiO2** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, type `Substrate - Si` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Height** text field, type `t_s`.

4 Locate the **Position** section. In the **y** text field, type `-(t_lil*3)-(t_hil*3)-t_i-t_s`.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

6 In the **New Cumulative Selection** dialog box, type `Si` in the **Name** text field.

7 Click **OK**.

Bottom PML - Si

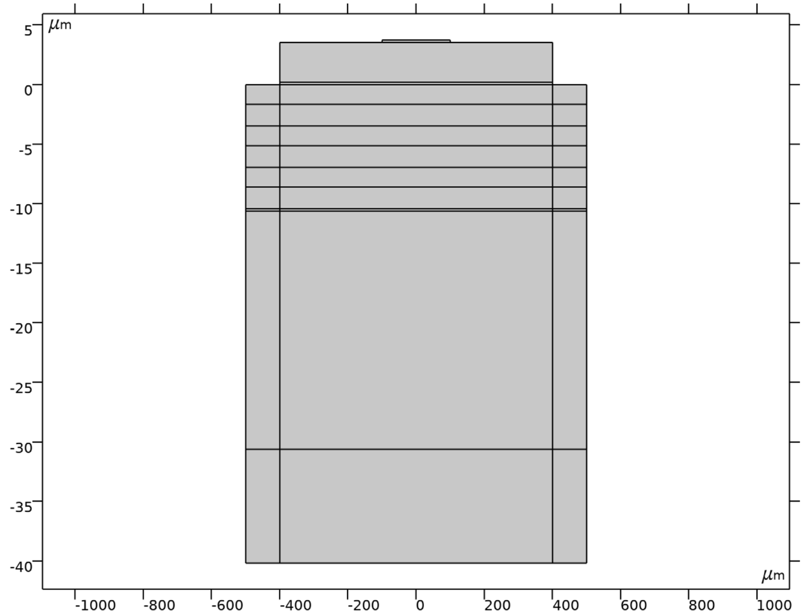
1 Right-click **Substrate - Si** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, type `Bottom PML - Si` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Height** text field, type `lambda_Si`.

4 Locate the **Position** section. In the **y** text field, type `-(t_lil*3)-(t_hil*3)-t_i-t_s-lambda_Si`.

5 In the **Geometry** toolbar, click **Build All**.



Create selections for the PML and fixed boundary condition.

DEFINITIONS

Not PML

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

2 In the **Settings** window for **Box**, type `Not PML` in the **Label** text field.

3 Locate the **Box Limits** section. In the **x minimum** text field, type `-w_ar`.

4 In the **x maximum** text field, type `w_ar`.

5 In the **y minimum** text field, type `-(t_hil*3)-(t_lil*3)-t_i-t_s/2`.

PML

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

2 In the **Settings** window for **Complement**, type `PML` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Not PML** in the **Selections to invert** list.

5 Click **OK**.

Left fixed B.C.

1 In the **Definitions** toolbar, click **Box**.
2 In the **Settings** window for **Box**, type `Left fixed B.C.` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Box Limits** section. In the **x maximum** text field, type `-(w+w_pe)/4`.
5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Right fixed B.C.

1 Right-click **Left fixed B.C.** and choose **Duplicate**.
2 In the **Settings** window for **Box**, type `Right fixed B.C.` in the **Label** text field.
3 Locate the **Box Limits** section. In the **x minimum** text field, type `(w+w_pe)/4`.
4 In the **x maximum** text field, type `Inf`.

Bottom fixed B.C.

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Bottom fixed B.C.` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundaries 2, 21, and 49 only.

Fixed B.C.

1 In the **Definitions** toolbar, click **Union**.
2 In the **Settings** window for **Union**, type `Fixed B.C.` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
5 In the **Add** dialog box, in the **Selections to add** list, choose **Left fixed B.C.**, **Right fixed B.C.**, and **Bottom fixed B.C.**.
6 Click **OK**.

Create the PMLs.

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.
2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **PML**.

Before adding material properties, set up the physics settings, so that the required properties will be highlighted when adding materials. In this 2D model, enter the out-of-plane thickness into each physics interface, so that lumped parameters can be computed appropriately. Use the selections made earlier for the physics selections. For Solid Mechanics: add damping subnodes and fixed boundary condition.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Thickness** section.

3 In the d text field, type `w_ar`.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

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 **Damping type** list, choose **Isotropic loss factor**.

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Piezoelectric Material 1**.

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

3 From the **Selection** list, choose **Piezo - ZnO**.

Mechanical Damping 1

In the **Physics** toolbar, click **Attributes** and choose **Mechanical Damping**.

Fixed Constraint 1

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

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

3 From the **Selection** list, choose **Fixed B.C.**.

For Electrostatics: only the domain surrounded by electrodes (the piezo domain) needs to be selected; remember to enter the out-of-plane thickness. Use the Terminal boundary condition (not the Electric Potential boundary condition) for the excitation port, so that lumped electrical parameters will be computed automatically.

ELECTROSTATICS (ES)

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

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

3 From the **Selection** list, choose **Piezo - ZnO**.

4 Locate the **Thickness** section. In the d text field, type `w_ar`.

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 **Bottom electrode - Al**.

Terminal 1

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

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

3 From the **Selection** list, choose **Top electrode - Al**.

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

The domain and physics selections of the Piezoelectric Effect multiphysics coupling should be set up automatically.

MULTIPHYSICS

Piezoelectric Effect 1 (pze1)

Add material properties from the COMSOL Piezoelectric, MEMS, and Built-in material folders as an initial template. Then enter the available data from the reference paper using the parameters prepared earlier under **Parameters 2 - Material properties**.

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 **Piezoelectric>Zinc Oxide**.

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

MATERIALS

Zinc Oxide (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Piezo - ZnO**.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rho_ZnO	kg/m³	Basic
Loss factor for elasticity matrix cE	eta_cE_iso ; eta_cEii = eta_cE_iso, eta_cEij = 0	eta0	1	Stress-charge form

ADD MATERIAL

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

2 In the tree, select **MEMS>Metals>Al - Aluminum**.

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

MATERIALS

Al - Aluminum (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Al**.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Isotropic structural loss factor	eta_s	eta0	1	Basic
Density	rho	rho_Al	kg/m³	Basic
Young's modulus	E	E_Al	Pa	Young's modulus and Poisson's ratio

ADD MATERIAL

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

2 In the tree, select **MEMS>Insulators>SiO2 - Silicon oxide**.

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

MATERIALS

SiO2 - Silicon oxide (mat3)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **SiO2**.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Isotropic structural loss factor	eta_s	`eta0`	1	Basic
Density	rho	`rho_SiO2`	kg/m³	Basic
Young's modulus	E	`E_SiO2`	Pa	Young's modulus and Poisson's ratio

ADD MATERIAL

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

2 In the tree, select **MEMS>Semiconductors>Si - Silicon (single-crystal, isotropic)**.

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

MATERIALS

Si - Silicon (single-crystal, isotropic) (mat4)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Si**.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Isotropic structural loss factor	eta_s	`eta0`	1	Basic
Density	rho	`rho_Si`	kg/m³	Basic
Young's modulus	E	`E_Si`	Pa	Young's modulus and Poisson's ratio

ADD MATERIAL

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

2 In the tree, select **Built-in>Molybdenum**.

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

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

MATERIALS

Molybdenum (mat5)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Mo**.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Isotropic structural loss factor	eta_s	`eta0`	1	Basic
Density	rho	`rho_Mo`	kg/m^3	Basic
Young's modulus	E	`E_Mo`	Pa	Young's modulus and Poisson's ratio

To save time and file size, a relatively coarse mesh will be used, in particular in the horizontal direction. Thus only the main lower modes will be resolved in this model. The same approach was taken in the reference paper.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

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

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

4 From the **Selection** list, choose **Top electrode - Al**.

5 Click to expand the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

6 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 45 only.

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

4 In the **Number of elements** text field, type `16`.

Distribution 2

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

2 Select Boundary 43 only.

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

4 In the **Number of elements** text field, type `1`.

Edge 1

1 In the **Mesh** toolbar, click **Edge**.

2 Select Boundaries 42 and 47 only.

3 In the **Settings** window for **Edge**, click to expand the **Control Entities** section.

4 Clear the **Smooth across removed control entities** check box.

Distribution 1

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

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

3 In the list, select **42**.

4 Click **Remove from Selection**.

5 Select Boundary 47 only.

6 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

7 In the **Number of elements** text field, type 16.

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

9 From the **Growth rate** list, choose **Exponential**.

Distribution 2

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

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

3 Click **Clear Selection**.

4 Select Boundary 42 only.

5 Locate the **Distribution** section. Select the **Reverse direction** check box.

Copy Edge 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations> Copy Edge**.

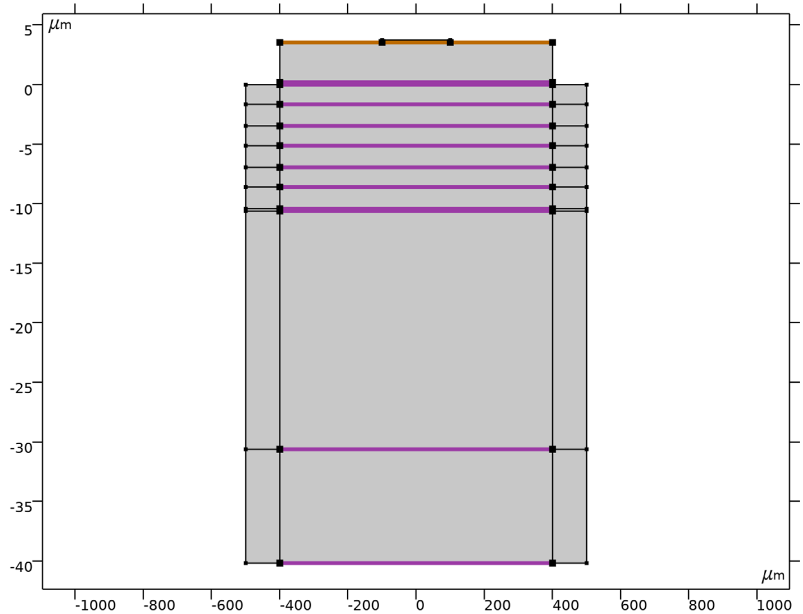
2 Select Boundaries 42, 44, and 47 only.

3 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Boundaries 21, 23, 25, 27, 29, 31, 33, 35, 37, 39, and 41 only.

6 Click to clear the **Activate Selection** toggle button.



Mapped 2

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Control Entities** section.

3 Clear the **Smooth across removed control entities** check box.

4 Locate the **Reduce Element Skewness** section. Select the **Adjust edge mesh** check box.

Distribution 1

1 Right-click **Mapped 2** and choose **Distribution**.

2 Select Boundary 40 only.

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

4 In the **Number of elements** text field, type 8.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 2** and choose **Distribution**.

2 Select Boundaries 5 and 38 only.

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

4 In the **Number of elements** text field, type 1.

Distribution 3

1 Right-click **Mapped 2** and choose **Distribution**.

2 Select Boundaries 7, 9, 11, 13, 15, 17, 19, and 67 only.

Distribution 4

1 Right-click **Mapped 2** and choose **Distribution**.

2 Select Boundaries 3, 22, 50, and 70 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 16.

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

7 From the **Growth rate** list, choose **Exponential**.

8 Select the **Reverse direction** check box.

Distribution 5

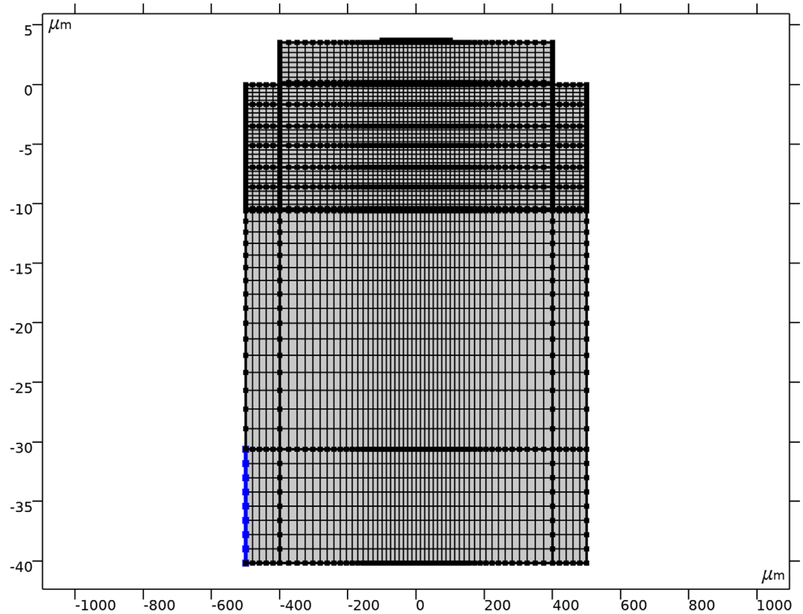
1 Right-click **Mapped 2** and choose **Distribution**.

2 Select Boundary 1 only.

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

4 In the **Number of elements** text field, type 8.

5 Click **Build All**.



Use the eigenfrequency study to look for the lower modes around 870 MHz.

STUDY 1 - MODES

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

2 In the **Settings** window for **Study**, type `Study 1 - Modes` in the **Label** text field.

Step 1: Eigenfrequency

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

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

3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type `4`.

5 Find the **Search region** subsection. From the **Unit** list, choose **MHz**.

6 In the **Search for eigenfrequencies around** text field, type `870`.

7 From the **Eigenfrequency search method around shift** list, choose **Larger real part**.

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

RESULTS

Mode Shape (solid)

Adjust the deformation scaling for better visualization of the mode shape. Then zoom in to the top part of the structure.

1 In the **Model Builder** window, expand the **Mode Shape (solid)** node.

Deformation

1 In the **Model Builder** window, expand the **Results>Mode Shape (solid)>Surface 1** node, then click **Deformation**.

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

3 Select the **Scale factor** check box.

4 In the associated text field, type `2e12`.

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

MATERIALS

Molybdenum (mat5)

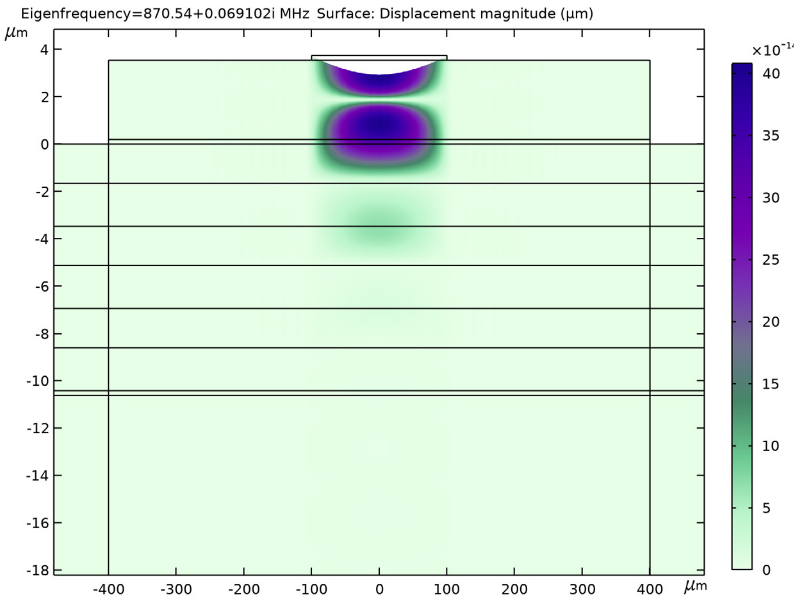
1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Molybdenum (mat5)**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 Click **Zoom to Selection**.

RESULTS

Mode Shape (solid)



The acoustic mirror effectively confines the mode energy at the top of the structure as expected.

Add a study for the frequency response. The frequency list is tailored to show more details near the main resonance and less elsewhere.

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>Frequency Domain**.

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

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

STUDY 2

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 From the **Frequency unit** list, choose **MHz**.

3 In the **Frequencies** text field, type `range(500,50,800) range(810,10,850) range(860,2,870) range(870.1,0.05,870.9) range(871,1,910) 920 930 940 range(950,50,1200)`.

4 In the **Model Builder** window, click **Study 2**.

5 In the **Settings** window for **Study**, type `Study 2 - Frequency Response` in the **Label** text field.

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

Take a look at the electric potential solution at the main resonance.

RESULTS

Electric Potential (es) 1

1 In the **Model Builder** window, under **Results** click **Electric Potential (es) 1**.

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

3 From the **Parameter value (freq (MHz))** list, choose **870.55**.

4 In the **Electric Potential (es) 1** toolbar, click **Plot**.

MATERIALS

Molybdenum (mat5)

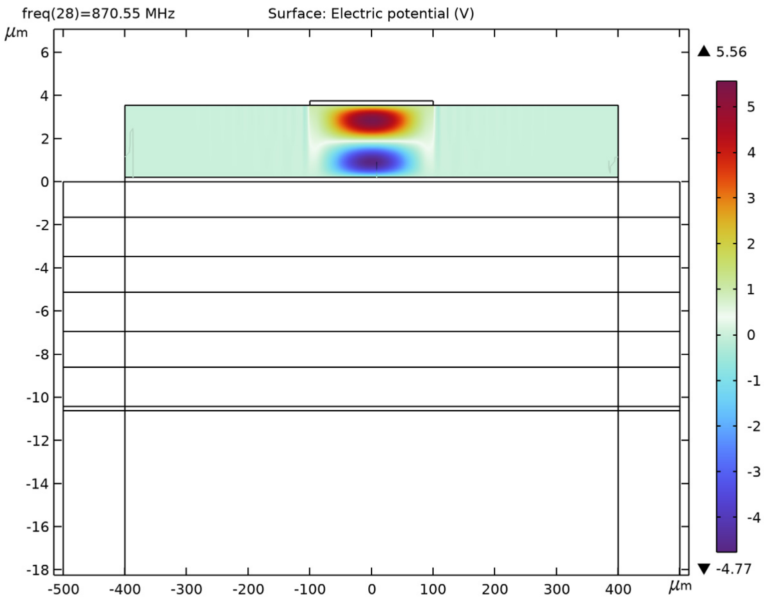
1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Molybdenum (mat5)**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 Click **Zoom to Selection**.

RESULTS

Electric Potential (es) 1



Add plots of the impedance and displacement to be compared with Fig.4 and 5 in the reference paper.

Log10|Z| - Fig.4

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Log10|Z| - Fig.4` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Response/ Solution 2 (sol2)**.
4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.
5 Locate the **Legend** section. Clear the **Show legends** check box.

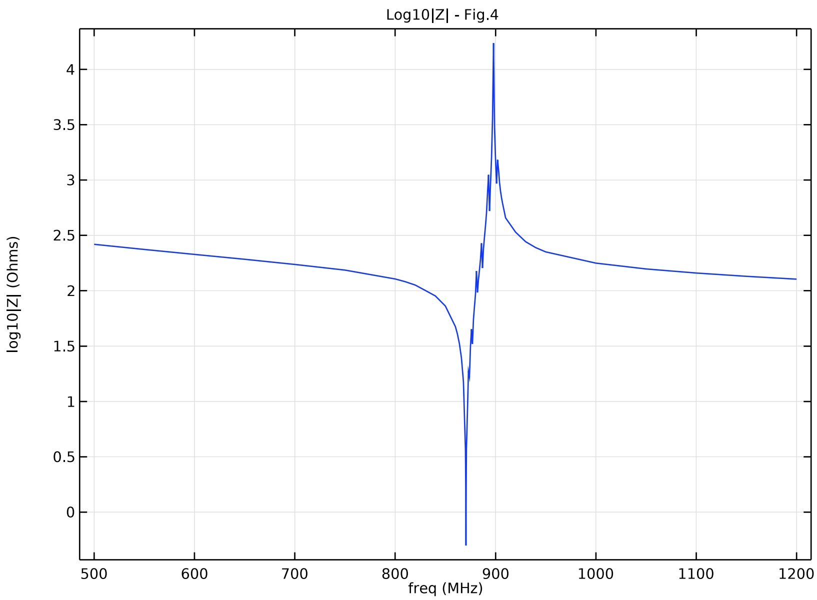
Global 1

1 Right-click **Log10|Z| - Fig.4** 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(1/es.Y11)/1[ohm])`		`log10|Z| (Ohms)`

4 In the **Log10|Z| - Fig.4** toolbar, click **Plot**.



Displacement profile - Fig.5(a)

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

2 In the **Settings** window for **2D Plot Group**, type `Displacement profile - Fig.5(a)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Response/ Solution 2 (sol2)**.

4 From the **Parameter value (freq (MHz))** list, choose **870.55**.

5 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

Surface 1

1 Right-click **Displacement profile - Fig.5(a)** and choose **Surface**.

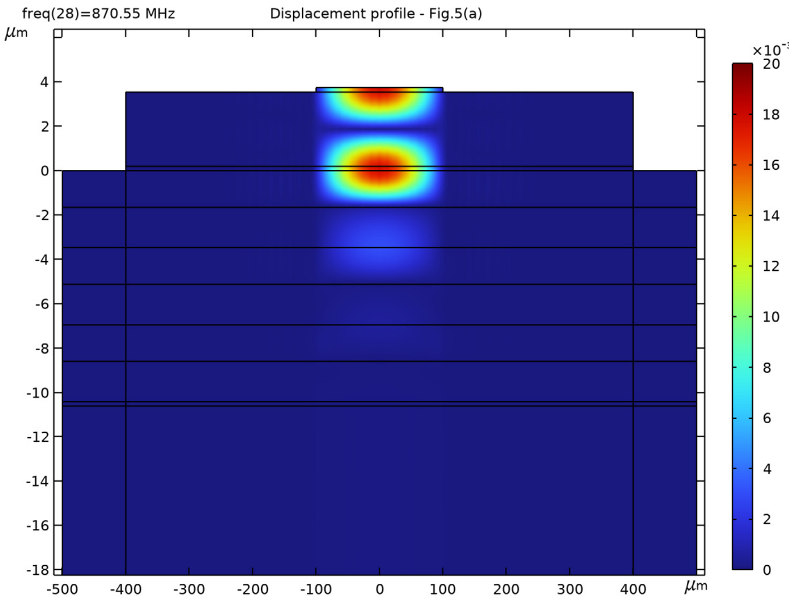
2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `sqrt(abs(u)^2+abs(v)^2)`.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Maximum** text field, type `0.02`.

6 In the **Displacement profile - Fig.5(a)** toolbar, click **Plot**.



Cut Line 2D 1

1 In the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2 - Frequency Response/Solution 2 (sol2)**.

4 Locate the **Line Data** section. In row **Point 1**, set **y** to `4[um]`.

5 In row **Point 2**, set **y** to `-20[um]`.

Y Displacement - Fig.5(b)

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

2 In the **Settings** window for **1D Plot Group**, type `Y Displacement - Fig.5(b)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 2D 1**.

4 From the **Parameter selection (freq)** list, choose **From list**.

5 In the **Parameter values (freq (MHz))** list, select **870.55**.

6 Locate the **Title** section. From the **Title type** list, choose **Label**.

7 Locate the **Axis** section. Select the **Manual axis limits** check box.

8 In the **x minimum** text field, type `-0.02`.

9 In the **x maximum** text field, type `0.02`.

10 In the **y minimum** text field, type `-20.5`.

11 In the **y maximum** text field, type `4.5`.

Line Graph 1

1 Right-click **Y Displacement - Fig.5(b)** and choose **Line Graph**.

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

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

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

5 In the **Expression** text field, type `-imag(v)`.

6 Select the **Description** check box.

7 In the associated text field, type `Displacement, Y component`.

8 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Red**.

9 In the **Width** text field, type `2`.

Line Segments 1

1 In the **Model Builder** window, right-click **Y Displacement - Fig.5(b)** and choose **Line Segments**.

2 In the **Settings** window for **Line Segments**, locate the **x-Coordinates** section.

3 In the table, enter the following settings:

Expression	Unit	Description
-0.02	1	
0.02	1	
0.02	1	
-0.02	1	
-0.02	1	
0.02	1	
0.02	1	
-0.02	1	
-0.02	1	
0.02	1	
0.02	1	

Expression	Unit	Description
-0.02	1	
-0.02	1	
0.02	1	
0.02	1	
-0.02	1	
-0.02	1	
0.02	1	
0.02	1	
-0.02	1	
-0.02	1	
0.02	1	

4 Locate the **y-Coordinates** section. In the table, enter the following settings:

Expression	Unit	Description
2*t_e+t_pe	µm	
2*t_e+t_pe	µm	
t_e+t_pe	µm	
t_e+t_pe	µm	
t_e	µm	Electrode thickness
t_e	µm	Electrode thickness
0	1	
0	1	
-t_lil	µm	
-t_lil	µm	
-t_lil-t_hil	µm	
-t_lil-t_hil	µm	
-2*t_lil-t_hil	µm	
-2*t_lil-t_hil	µm	
-2*t_lil-2*t_hil	µm	
-2*t_lil-2*t_hil	µm	
-3*t_lil-2*t_hil	µm	
-3*t_lil-2*t_hil	µm	
-3*t_lil-3*t_hil	µm	
-3*t_lil-3*t_hil	µm	

Expression	Unit	Description
-3*t_lil-3*t_hil-t_i	µm	
-3*t_lil-3*t_hil-t_i	µm	

5 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Black**.

Y Displacement - Fig.5(b)

In the **Model Builder** window, click **Y Displacement - Fig.5(b)**.

Table Annotation 1

1 In the **Y Displacement - Fig.5(b)** toolbar, click **More Plots** and choose **Table Annotation**.

2 In the **Settings** window for **Table Annotation**, locate the **Data** section.

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
-0.02	2	ZnO
-0.02	-0.85	SiO2
-0.02	-2.6	Mo
-0.02	-4.3	SiO2
-0.02	-6	Mo
-0.02	-7.8	SiO2
-0.02	-9.5	Mo
-0.02	-14	Substrate

5 Locate the **Coloring and Style** section. Clear the **Show point** check box.

6 From the **Anchor point** list, choose **Middle left**.

7 In the **Y Displacement - Fig.5(b)** toolbar, click **Plot**.