Stationary Analysis of a Biased Resonator — 2D

Introduction

Silicon micromechanical resonators have long been used for designing sensors and are now becoming increasingly important as oscillators in the consumer electronics market. In this sequence of models, a surface micromachined MEMS resonator, designed as part of a micromechanical filter, is analyzed in detail. The resonator is based on that developed in Ref. 1.

This model performs a stationary analysis of the resonator, with an applied DC bias. It is used as a basis for all the subsequent analyses.

Model Definition

Ref. 1 describes a polysilicon resonator, which is manufactured through a surface micromachining process. The details of this process are outlined in the Stationary Analysis of a Biased Resonator — 3D documentation. For this 2D study, a simplified version of the 3D geometry is considered. For simplicity, the resonator is modeled as a 2 µm thick rectangular beam with a length of 45 µm. A Fixed Constraint boundary is applied to each end of the resonator to act as the anchor points at which the resonator is attached to the substrate wafer. The wafer substrate is not explicitly modeled, instead only a 0.1985 µm thick air gap between the resonator and the substrate is included. The effects of the driving electrode are included using Electric Potential boundary conditions applied directly to the underside of the air gap, as shown in Figure 1.

Note that although the structure has a plane of symmetry, which vertically bisects the device, we do not use a symmetry boundary condition. A subsequent model considers the normal modes of the structure, and a symmetry condition eliminates the anti-symmetric modes from this analysis.

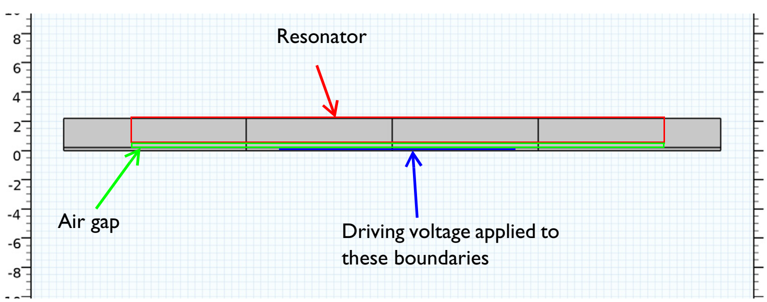

Figure 1: The model geometry. The rectangular resonator and the air gap are highlighted in red and green outline, respectively. The boundaries to which the driving voltage is applied are

highlighted in blue. Note that the vertical dividing lines are not part of the physical geometry of the resonator, but are included to allow a suitable swept mesh to be easily created.

In operation the silicon resonator is grounded (using the Domain Terminal feature) and a driving electrode applies an electric potential to the central portion of the air gap, as shown in Figure 1. Typically a DC bias of 35 V is applied in normal operation of the device. In this model the deformation of the structure is computed with the applied DC bias.

ELECTROMECHANICAL FORCES

Within a vacuum or other medium, forces between charged bodies can be computed on the assumption that a fictitious state of stress exists within the field. The Electromagnetic or Maxwell stress tensor can be used to compute the induced stresses in a material as a result of an electric field as well as surface forces acting on bodies in air or vacuum. Within a material, COMSOL Multiphysics uses the following form of the stress tensor $T_{\mathrm{EM},S}$, which is appropriate for isotropic materials (Ref. 2):

$$T_{EM,S} = -\frac{1}{2}(\mathbf{E}\cdot\mathbf{D}+a_2\mathbf{E}\cdot\mathbf{E})I+\mathbf{E}\mathbf{D}^T+\frac{1}{2}(a_2-a_1)\mathbf{E}\mathbf{E}^T$$

where $\mathbf{E}$ is the electric field, $\mathbf{D}$ is the electric displacement field, I is the identity tensor, and ε_0 is the permittivity of free space and a_1 and a_2 are material parameters that specify the electrostrictive properties of the material (for this device, assume $a_1 = a_2 = 0$ because the field is in any case very low within the material). This additional stress is applied to the material by the electromechanical solid node. Note that mechanical stresses are usually induced in the material as a result of the net forces acting on the surfaces, in addition to the stress induced by the electric field.

The forces on the surfaces of a solid body can be computed by applying a similar stress term within the vacuum of the form:

$$T_{\mathrm{EM},V} = -\frac{1}{2}(\mathbf{E}\cdot\mathbf{D})I+\mathbf{E}\mathbf{D}^T$$

A net force on the surface typically results from the discontinuity of the stress tensor at the interface. However, since it is undesirable to apply a stress term throughout the vacuum, the force is only available on the surface of solid bodies, via the electromechanical interface node. The surface force is given by:

$$\mathbf{n}_1 T_{\mathrm{EM},V} = -\left(\frac{1}{2}\mathbf{E}\cdot\mathbf{D}\right)\mathbf{n}_1+(\mathbf{n}_1\cdot\mathbf{E})\mathbf{D}$$

where $\mathbf{n}_1$ is the surface normal, pointing out from the mechanical body.

Results and Discussion

Figure 2 shows the y displacement of the structure with an applied DC bias. As expected the structural displacement is maximal at the center of the geometry. The maximum displacement is 11.2 nm.

The electric potential contours are shown in Figure 3. The fringing fields extend approximately 1 μm into the gap either side of the driving electrode. Note that the fringing fields are not well resolved due to the structure of the swept mesh. In order to investigate these fields the mesh must be refined on either side of the electrode. Also note that the silicon is assumed to be a perfect conductor. Although the author's of Ref. 1 do not explicitly give the doping in the polysilicon, it is likely that this assumption is relatively poor given the estimated depletion region width of approximately 0.7 μm that is quoted. This model could be extended to include the effects of semiconductor transport and an improvement on this assumption could be made by adding the electric currents which are induced inside the resonator.

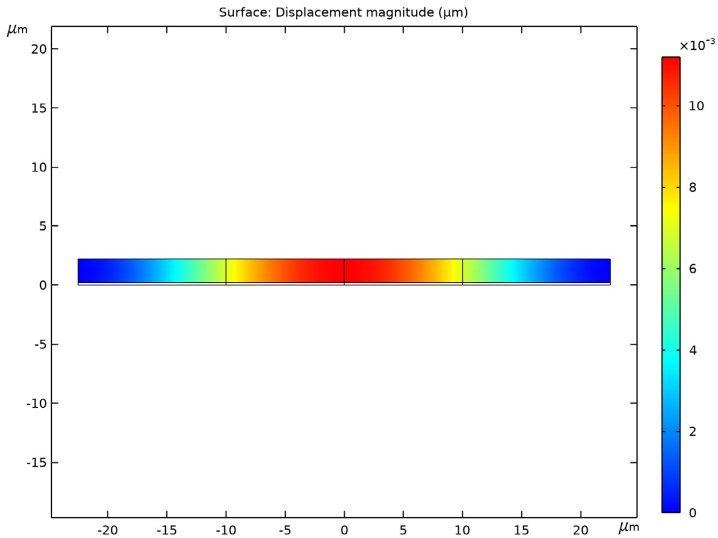

Figure 2: The y-displacement of the resonator as a function of position. The maximum displacement occurs in the center of the resonator, immediately over the biasing electrode.

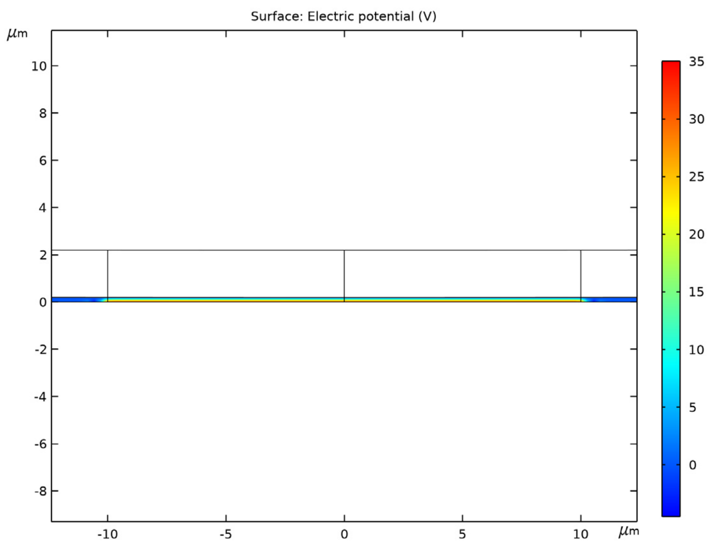

Figure 3: Electric potential contours in the gap between the grounded resonator and the biased driving electrode.

References

1. F.D. Bannon III, J.R. Clark, and C.T.-C. Nguyen, "High-Q HF Microelectromechanical Filters," *IEEE Journal of Solid State Circuits*, vol. 35, no. 4, pp. 512–526, 2000.

2. J.A. Stratton, *Electromagnetic Theory*, McGraw-Hill, New York, 1941.

Application Library path: `MEMS_Module/Actuators/biased_resonator_2d_basic`

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**.

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

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
Vdc	35[V]	35 V	DC bias voltage

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 **µm**.

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 `22.5`.

4 In the **Height** text field, type `2`.

5 Locate the **Position** section. In the **x** text field, type `-22.5`.

6 In the **y** text field, type `0.1985`.

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 `22.5`.

4 In the **Height** text field, type `0.1985`.

5 Locate the **Position** section. In the **x** text field, type `-22.5`.

Rectangle 3 (r3)

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 `10`.

4 In the **Height** text field, type `2.1985`.

5 Locate the **Position** section. In the **x** text field, type `-10`.

Mirror 1 (mir1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Click **Build All Objects**.

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

Add materials to the model.

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 **MEMS>Semiconductors>Si - Polycrystalline silicon**.

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

5 In the tree, select **Built-in>Air**.

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

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

MATERIALS

Air (mat2)

1 Select Domains 1, 3, 5, and 7 only.

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

3 Click **Create Selection**.

4 In the **Create Selection** dialog box, type `Air` in the **Selection name** text field.

5 Click **OK**.

6 In the **Settings** window for **Material**, click to expand the **Material Properties** section.

SOLID MECHANICS (SOLID)

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

2 Select Domains 2, 4, 6, and 8 only.

ELECTROSTATICS (ES)

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

Charge Conservation 2

1 In the **Physics** toolbar, click **Domains** and choose **Charge Conservation**.

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

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

DEFINITIONS

Deforming Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Moving Mesh** click **Deforming Domain 1**.

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

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

Set up the solid mechanics and electrostatics boundary conditions.

SOLID MECHANICS (SOLID)

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

Fixed Constraint 1

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

Apply a **Fixed Constraint** to both ends of the resonator.

2 Select Boundaries 3 and 22 only.

ELECTROSTATICS (ES)

With the assumption that the silicon membrane is a good conductor, use the Domain Terminal feature to ground the Si domains. Note: The Domain Terminal feature will be very handy for a conducting domain with a complex shape and many exterior boundaries - instead of selecting all the boundaries to set up the Ground, Terminal, or Electric Potential boundary condition, we only need to select the domain to specify the Domain Terminal with the same effect. In addition, the computation load is reduced, because the electrostatic degrees of freedom within the Domain Terminal do not need to be solved for.

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

Terminal 1

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

2 Select Domains 2, 4, 6, and 8 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 `0`.

Electric Potential 1

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

Set the bias voltage on the driving electrode with the **Electric Potential** feature.

2 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

3 In the V_0 text field, type `Vdc`.

4 Select Boundaries 7 and 12 only.

Modify the default mesh settings to suit the model geometry.

A mapped mesh allows good resolution of the small air gap between the driving electrode and the resonator.

MESH 1

Mapped 1

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

Distribution 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `5 10 15 20` in the **Selection** text field.

5 Click **OK**.

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

7 In the **Number of elements** text field, type `15`.

Distribution 2

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1 3` in the **Selection** text field.

5 Click **OK**.

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

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

8 Click **Build All**.

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

STATIONARY

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

2 In the **Rename Study** dialog box, type `Stationary` in the **New label** text field.

3 Click **OK**.

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

RESULTS

Biased Displacement

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

2 In the **Biased Displacement** toolbar, click **Plot**.

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

Compare the resulting plot with that in Figure 2.

Selection 1

1 In the **Model Builder** window, expand the **Results>Electric Potential (es)** node.

2 Right-click **Surface 1** and choose **Selection**.

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

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

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

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

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

Compare the resulting plot with that in Figure 3.