Manufacturing Variation Effects in a Micromachined Comb-Drive Tuning Fork Rate Gyroscope

Introduction

This tutorial model is kindly provided by Dr. James Ransley at Veryst Engineering, LLC. It continues from the base model A Micromachined Comb-Drive Tuning Fork Rate Gyroscope, which is also provided by Dr. Ransley. The model demonstrates how to accurately compute the effects of manufacturing variations of MEMS devices without the need of very fine mesh. This highly efficient modeling approach is based on the unique Deformed Geometry functionality of COMSOL Multiphysics, which implements the device shape change due to fabrication imperfections using the same mesh, thus eliminating the error introduced if different meshes were to be used for different geometries. The device is loosely based on Ref. 1.

Model Definition

In this tutorial, manufacturing variations are added to the entire device layer which simulate the effect of an over-etch causing critical dimension (CD) variation, device layer thickness variations and a sidewall tilt parameterized by the steepest angle to the vertical (theta) and an in-plane angle (phi). See Figure 1 for the three types of variations. In this model, the variations are spatially uniform for simplicity. However, it is straightforward to make these variations a function of space within the device, which can also lead to systematic variations in the operation of the gyroscope, and to combine the variations arbitrarily. To implement a space-wise variation, the variable definitions can be modified so that they are a function of the model coordinates (an imported interpolation function can also be used). It is also straightforward to extend these variations to the anchor layer, although that is not done here for simplicity.

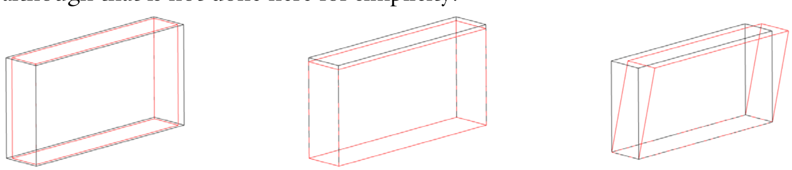


Figure 1: Types of manufacturing variability implemented in this example: Over-etch (left), Thickness (middle), and Sidewall (right).

These variations are implemented using simple equations added with the Deformed Geometry interface, which are then solved alongside the equations for gyroscope motion. It is simplest to implement these deformations as normal displacements to the vertical walls of the structure (for the sidewall and over-etch) and the upper surface of the structure (for the thickness variation). As implemented here, the deviations assume that the base of the device layer is at a z-coordinate of zero. Note also that when solving the model, the deformed geometry only needs to be added during the stationary solution of the linearization point for the model; this is reflected in the study settings in the step-by-step instructions (see Modeling Instructions).

See the documentation of the base model, A Micromachined Comb-Drive Tuning Fork Rate Gyroscope, for additional details.

Results and Discussion

Table 1 shows the frequency sensitivity for the drive and sense modes. This is important as the modal overlap determines the sensitivity of the device in a mode-split gyroscope such as this one. It is clear that the mode spacing is strongly affected by the CD control with a 50 nm over-etch (resulting in 100 nm of CD variation, which is relatively poor process control) causing large changes in the spacing of the modes. The drive mode is relatively unaffected by thickness variations and sidewall, as might be expected. The sense mode is significantly affected by the thickness – since it involves out of plane motion.

TABLE 1: SHIFTS OF THE DRIVE-MODE AND SENSE-MODE FREQUENCIES DUE TO MANUFACTURING VARIATIONS.

	Drive mode	Sense mode
Mode frequency (Hz)	38262	41129
Shift due to 50 nm over-etch (Hz)	-1272	-373
Shift due to 100 nm thickness change (Hz)	1	138
Shift due to 0.5 deg sidewall (Hz)	-3	14

The frequency domain study shows how to calculate the effect of sidewall for the case of a uniform tilt of magnitude 0.5 degree at an azimuthal angle of 45 degrees to the die edges. This case shows the inherent challenge of gyroscope system design, as a large quadrature signal shows up as a capacitance close to 90 degrees out of phase from the real signal due to the rotation. This signal is normally filtered out by the sense electronics, however, it is still common for some fraction to appear as an offset to the gyroscope output. A careful analysis of the data would consider the precise phase shift between the drive and sense modes and extract the rate signal at the appropriate phase. For the purpose of this example, we simply note that an applied rate results in a similar change to the real part of the signal (902 aF-879 aF=23 aF ~ 23 aF), and that the gyroscope offset is equivalent to approximately 3800 deg/s (=879 aF/(23 aF/100 deg/s)).

TABLE 2: COMPUTED AC SENSE CAPACITANCE WITH AND WITHOUT SIDEWALL VARIATION.

	No rotation		100 deg/s rotation	
	Real part	Imaginary part	Real part	Imaginary part
AC capacitance for ideal device (aF)	0	0	23	-0.4
AC capacitance for device with 0.5 deg sidewall (aF)	879	10585	902	10583

The importance of tightly controlled manufacturing variability and design for manufacture is apparent from this example.

Figure 2 and Figure 3 show the sense-mode out-of-plane seesaw motion appearing in the drive-mode shape once the sidewall tilt is added. This motion is responsible for the large imaginary AC capacitance shown in Table 2.

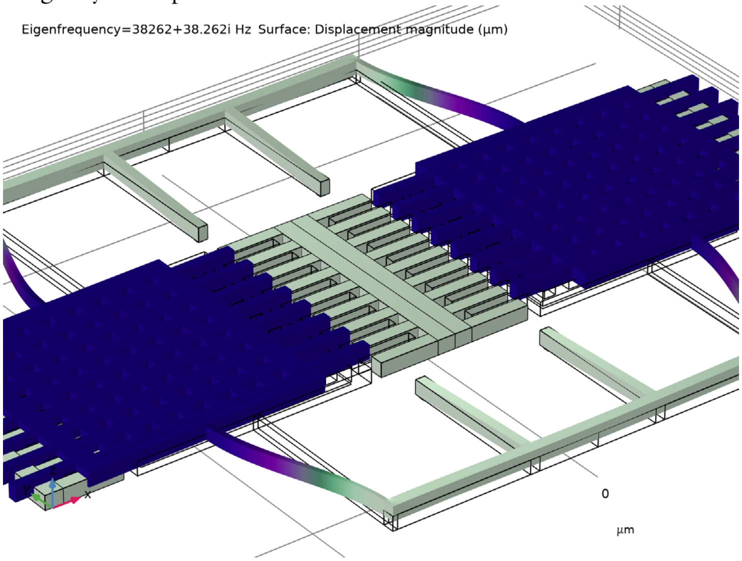


Figure 2: Ideal drive-mode shape with 20X scaling in the out-of-plane direction.

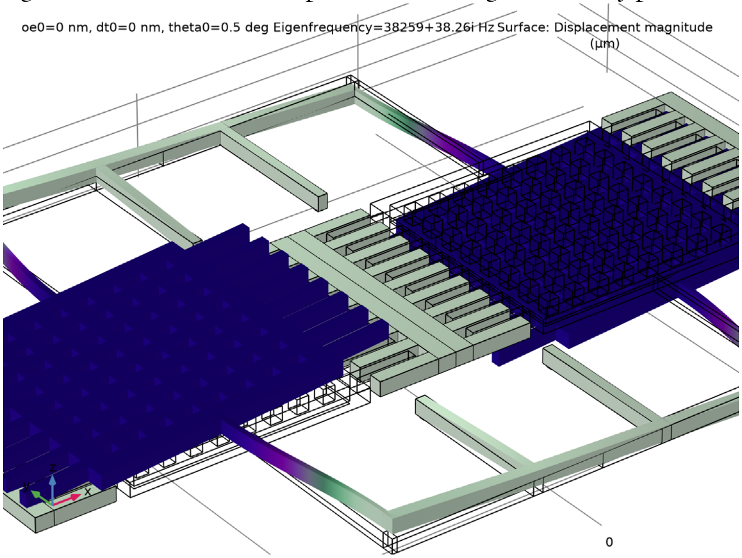


Figure 3: Drive-mode shape with 0.5 deg sidewall and 20X out-of-plane scaling.

Figure 4 and Figure 5 show that the out-of-plane seesaw motion also appears in the real part of the displacement during operation once the sidewall tilt is added. This motion is responsible for the large real offset of the AC capacitance shown in Table 2.

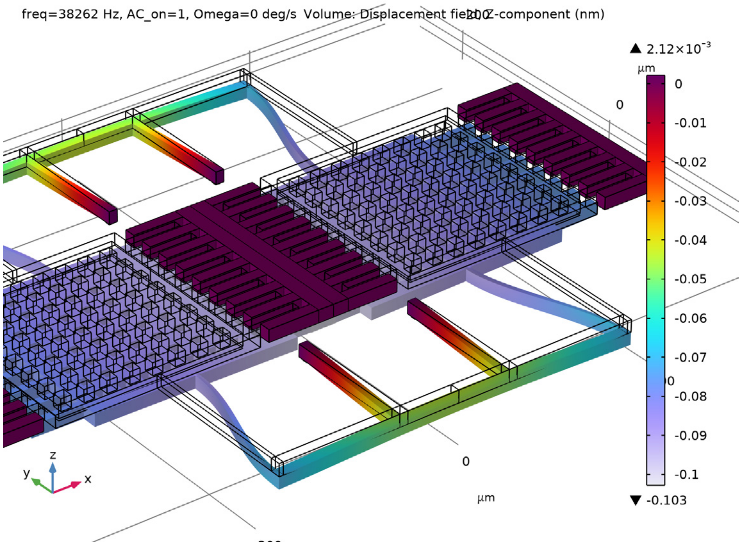


Figure 4: Displacement of ideal device during operation with no rotation.

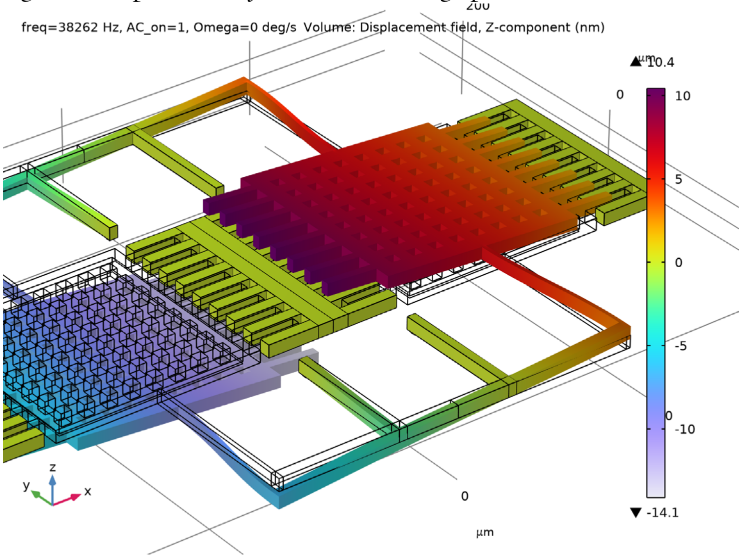


Figure 5: Displacement of device with side wall during operation with no rotation.

Figure 6 shows the shape change implemented by the Deformed Geometry interface, using a sidewall with zero azimuth and a large tilt angle of 10 degrees as an exaggerated example.

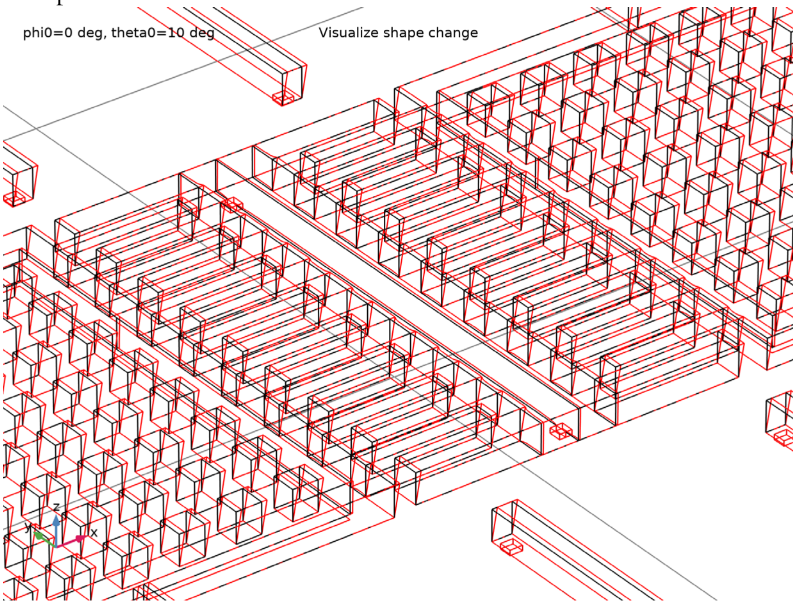


Figure 6: Visualizing the device shape with a sidewall of 10-degree tilt and zero azimuth. Black lines: the undeformed shape shown by the geometry frame. Red lines: device shape with the sidewall imperfection shown by the material frame. The tilt toward the +x direction is clearly seen.

Reference

1. J. Bernstein, S. Cho, A. T. King, A. Kourepenis, P. Maciel, and M. Weinberg, "*A micromachined comb-drive tuning fork rate gyroscope*," Proceedings IEEE Micro Electro Mechanical Systems, Fort Lauderdale, FL, USA, 1993, pp. 143–148.

Application Library path: `MEMS_Module/Sensors/comb_drive_tuning_fork_gyroscope_manufacturing_variation`

ROOT

Open the base model comb_drive_tuning_fork_gyroscope.

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **MEMS Module>Sensors>comb_drive_tuning_fork_gyroscope** in the tree.

3 Click **Open**.

Enter the parameters associated with the manufacturing variations, using a new Parameters node for clarity.

GLOBAL DEFINITIONS

Parameters 8 - Manufacturing variation

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

2 In the **Settings** window for **Parameters**, type `Parameters 8 - Manufacturing variation` in the **Label** text field.

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

Name	Expression	Value	Description
theta0	0.5[deg]	0.0087266 rad	Sidewall tilt
phi0	45[deg]	0.7854 rad	Sidewall azimuth
dt0	0[um]	0 m	Thickness variation
oe0	0[um]	0 m	Over-etch

Similarly add the variables associated with the manufacturing variations, using a new Variables node. As implemented here, the deviations assume that the base of the beam layer is at a z-coordinate of zero. In this model, for simplicity, the variables do not depend on the spatial location, but it is straightforward to make them spatially varying by either entering appropriate formulas or using imported functions.

DEFINITIONS

Variables 9 - Manufacturing variation

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

2 In the **Settings** window for **Variables**, type `Variables 9 - Manufacturing variation` in the **Label** text field.

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

Name	Expression	Unit	Description
theta	theta0	rad	Sidewall theta variation as function of global coordinates
phi	phi0	rad	Sidewall phi variation as function of global coordinates
dX_sw	Zg*tan(theta)*cos(phi)	m	Change in X for sidewall variation
dY_sw	Zg*tan(theta)*sin(phi)	m	Change in Y for sidewall variation
sw	nXg*dX_sw+nYg*dY_sw	m	Sidewall normal displacement
delta_t_function	dt0	m	Thickness variation as function of global coordinates
dZ_t	delta_t_function	m	Change in Z for thickness variation
dt	nZg*dZ_t	m	Thickness normal displacement
oe	oe0	m	Over-etch variation as function of global coordinates

Add a **Deformed Geometry** branch to implement the shape change due to manufacturing variation. This approach allows accurate simulation of the effect of the variation, even with a relatively coarse mesh as in this model. The reason is that the **Deformed Geometry** branch keeps the mesh structure the same while changing the shape of the device according to the specified sidewall or over-etch, and so on. So relative changes in the computed quantities are typically captured accurately, even if the final result is not fully mesh-converged. Without using this approach, one would need a highly mesh-converged solution to see the true differences, since the mesh structure would change between the different shapes.

Use the **Free Deformation** feature to allow the volume mesh to deform in the entire beam layer. Choose the **Laplace** smoothing type. For simplicity, the anchor layer is not deformed (assuming ideal shape) in this model.

After adding the **Deformed Geometry** branch, remember to clear the check boxes for the existing studies so that it does not affect those studies.

COMPONENT 1 (COMP1)

Deforming Domain 1 - Beam Layer

1 In the **Physics** toolbar, click **Deformed Geometry** and choose **Free Deformation**.

2 In the **Settings** window for **Deforming Domain**, type `Deforming Domain 1 - Beam Layer` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Box 2 - Entire Beam Layer**.

4 Locate the **Smoothing** section. From the **Mesh smoothing type** list, choose **Laplace**.

STUDY 1 - STATIONARY

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1 - Stationary** node, then 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 box for **Deformed geometry (Component 1)**.

STUDY 2 - PRESTRESSED EIGENFREQUENCY

1 In the **Model Builder** window, expand the **Study 2 - Prestressed Eigenfrequency** node, then 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 box for **Deformed geometry (Component 1)**.

Step 2: Eigenfrequency

1 In the **Model Builder** window, click **Step 2: Eigenfrequency**.

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

3 In the table, clear the **Solve for** check box for **Deformed geometry (Component 1)**.

STUDY 3 - PRESTRESSED FREQUENCY DOMAIN

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 3 - Prestressed Frequency Domain** node, then 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 box for **Deformed geometry (Component 1)**.

Step 2: Frequency Domain Perturbation

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

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

3 In the table, clear the **Solve for** check box for **Deformed geometry (Component 1)**.

To implement the shape changes from the manufacturing variation, it is simplest to implement them as normal displacements to the vertical walls (for the sidewall and over-etch) and the upper surfaces (for the thickness variation). We should also prevent vertical movement of the base plane (the bottom surfaces of the beam layer), and fix the mesh on the anchor faces. Define selections for these surfaces first.

DEFINITIONS

Box 21 - Vertical Surfaces

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

2 Right-click **Definitions** and choose **Selections>Box**.

3 In the **Settings** window for **Box**, type `Box 21 - Vertical Surfaces` in the **Label** text field.

4 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

5 Locate the **Box Limits** section. In the **z minimum** text field, type `+delta`.

6 In the **z maximum** text field, type `2*delta`.

7 Right-click **Box 21 - Vertical Surfaces** and choose **Duplicate**.

Box 22 - Top Surfaces

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Selections** click **Box 21 - Vertical Surfaces 1**.

2 In the **Settings** window for **Box**, type `Box 22 - Top Surfaces` in the **Label** text field.

3 Locate the **Box Limits** section. In the **z minimum** text field, type `t_beam-delta`.

4 In the **z maximum** text field, type `t_beam+delta`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Adjacent 1 - Exterior Surfaces of Beam Layer

1 In the **Definitions** toolbar, click **Adjacent**.
2 In the **Settings** window for **Adjacent**, type `Adjacent 1 - Exterior Surfaces of Beam Layer` in the **Label** text field.
3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.
4 In the **Add** dialog box, select **Box 2 - Entire Beam Layer** in the **Input selections** list.
5 Click **OK**.

Intersection 11 - Vertical Walls

1 In the **Definitions** toolbar, click **Intersection**.
2 In the **Settings** window for **Intersection**, type `Intersection 11 - Vertical Walls` 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 intersect**, click **Add**.
5 In the **Add** dialog box, in the **Selections to intersect** list, choose **Box 21 - Vertical Surfaces** and **Adjacent 1 - Exterior Surfaces of Beam Layer**.
6 Click **OK**.

Intersection 12 - Anchor Faces

1 In the **Definitions** toolbar, click **Intersection**.
2 In the **Settings** window for **Intersection**, type `Intersection 12 - Anchor Faces` 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 intersect**, click **Add**.
5 In the **Add** dialog box, in the **Selections to intersect** list, choose **Box 1 - Bottom of Beam** and **Extrude 2 - Anchors**.
6 Click **OK**.

Now use **Prescribed Normal Mesh Displacement** features to specify nontrivial normal displacements for the vertical walls and upper surfaces, and to prevent vertical movement of the base plane (the bottom surfaces of the beam layer). Use a **Prescribed Mesh Displacement** feature to fix the mesh on the anchor faces.

DEFORMED GEOMETRY

Prescribed Normal Mesh Displacement 1 - Vertical Walls

1 In the **Deformed Geometry** toolbar, click **Prescribed Normal Mesh Displacement**.

2 In the **Settings** window for **Prescribed Normal Mesh Displacement**, type `Prescribed Normal Mesh Displacement 1 - Vertical Walls` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Intersection 11 - Vertical Walls**.

4 Locate the **Prescribed Normal Mesh Displacement** section. In the $\mathbf{d}_n$ text field, type `-oe+sw`.

Prescribed Normal Mesh Displacement 2 - Top Surfaces

1 In the **Deformed Geometry** toolbar, click **Prescribed Normal Mesh Displacement**.

2 In the **Settings** window for **Prescribed Normal Mesh Displacement**, type `Prescribed Normal Mesh Displacement 2 - Top Surfaces` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Box 22 - Top Surfaces**.

4 Locate the **Prescribed Normal Mesh Displacement** section. In the $\mathbf{d}_n$ text field, type `dt`.

Prescribed Normal Mesh Displacement 3 - Base Plane

1 In the **Deformed Geometry** toolbar, click **Prescribed Normal Mesh Displacement**.

2 In the **Settings** window for **Prescribed Normal Mesh Displacement**, type `Prescribed Normal Mesh Displacement 3 - Base Plane` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Box 1 - Bottom of Beam**.

Prescribed Mesh Displacement 1 - Anchor Faces

1 In the **Deformed Geometry** toolbar, click **Prescribed Mesh Displacement**.

2 In the **Settings** window for **Prescribed Mesh Displacement**, type `Prescribed Mesh Displacement 1 - Anchor Faces` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Intersection 12 - Anchor Faces**.

Keep the same mesh as in the original model. Add new studies for manufacturing variation analysis. Copy study steps from existing studies. Remember the important point that the **Deformed Geometry** branch should be solved only in the stationary steps, and not in the harmonic perturbation steps.

First look at the effect on the eigenfrequencies. Lower the search frequency since over-etching weakens the springs, leading to lower resonant frequency.

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 **Empty Study**.

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

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

STUDY 4 - PRESTRESSED EIGENFREQUENCY WITH MANUFACTURING VARIATIONS

In the **Settings** window for **Study**, type `Study 4 - Prestressed Eigenfrequency with Manufacturing Variations` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
oe0 (Over-etch)	0 0 50	nm

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dt0 (Thickness variation)	0 100 0	nm

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta0 (Sidewall tilt)	0.5 0 0	deg

STUDY 2 - PRESTRESSED EIGENFREQUENCY

Step 1: Stationary

In the **Model Builder** window, under **Study 2 - Prestressed Eigenfrequency** right-click **Step 1: Stationary** and choose **Copy**.

STUDY 4 - PRESTRESSED EIGENFREQUENCY WITH MANUFACTURING VARIATIONS

In the **Model Builder** window, right-click **Study 4 - Prestressed Eigenfrequency with Manufacturing Variations** and choose **Paste Stationary**.

1 In the **Model Builder** window, under **Study 4 - Prestressed Eigenfrequency with Manufacturing Variations** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 In the table, select the **Solve for** check box for **Deformed geometry (Component 1)**.

STUDY 2 - PRESTRESSED EIGENFREQUENCY

Step 2: Eigenfrequency

In the **Model Builder** window, under **Study 2 - Prestressed Eigenfrequency** right-click **Step 2: Eigenfrequency** and choose **Copy**.

STUDY 4 - PRESTRESSED EIGENFREQUENCY WITH MANUFACTURING VARIATIONS

In the **Model Builder** window, right-click **Study 4 - Prestressed Eigenfrequency with Manufacturing Variations** and choose **Paste Eigenfrequency**.

1 In the **Model Builder** window, under **Study 4 - Prestressed Eigenfrequency with Manufacturing Variations** click **Step 2: Eigenfrequency**.
2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
3 In the **Desired number of eigenfrequencies** text field, type `2`.
4 In the **Search for eigenfrequencies around shift** text field, type `36600[Hz]`.
5 In the **Study** toolbar, click **Compute**.

Add an Evaluation Group to look at the shifts in the eigenfrequencies.

RESULTS

Evaluation Group 3 - Study 4 - Frequency Shifts due to Manufacturing Variations

1 In the **Results** toolbar, click **Evaluation Group**.
2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group 3 - Study 4 - Frequency Shifts due to Manufacturing Variations` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **None**.

Global Evaluation 1

1. Right-click **Evaluation Group 3 - Study 4 - Frequency Shifts due to Manufacturing Variations** and choose **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 4 - Prestressed Eigenfrequency with Manufacturing Variations/Parametric Solutions 1 (sol8)**.
4. From the **Eigenfrequency selection** list, choose **First**.
5. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`real(freq)-fd`	Hz	Drive mode freq shift

6. Right-click **Global Evaluation 1** and choose **Duplicate**.

Global Evaluation 2

1. In the **Model Builder** window, click **Global Evaluation 2**.
2. In the **Settings** window for **Global Evaluation**, locate the **Data** section.
3. From the **Eigenfrequency selection** list, choose **Last**.
4. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`real(freq)-fs`	Hz	Sense mode freq shift

5. In the **Evaluation Group 3 - Study 4 - Frequency Shifts due to Manufacturing Variations** toolbar, click **Evaluate**.

Now perform a prestressed frequency domain study for the case of a uniform sidewall of magnitude 0.5 degree and an azimuthal angle of 45 degrees to the die edges, as specified in the Parameters table.

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 **Empty Study**.
4. Click **Add Study** in the window toolbar.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 5 - PRESTRESSED FREQUENCY DOMAIN WITH MANUFACTURING VARIATIONS

In the **Settings** window for **Study**, type `Study 5 - Prestressed Frequency Domain with Manufacturing Variations` in the **Label** text field.

STUDY 3 - PRESTRESSED FREQUENCY DOMAIN

Step 1: Stationary

In the **Model Builder** window, under **Study 3 - Prestressed Frequency Domain** right-click **Step 1: Stationary** and choose **Copy**.

STUDY 5 - PRESTRESSED FREQUENCY DOMAIN WITH MANUFACTURING VARIATIONS

In the **Model Builder** window, right-click **Study 5 - Prestressed Frequency Domain with Manufacturing Variations** and choose **Paste Stationary**.

1 In the **Model Builder** window, under **Study 5 - Prestressed Frequency Domain with Manufacturing Variations** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 In the table, select the **Solve for** check box for **Deformed geometry (Component 1)**.

STUDY 3 - PRESTRESSED FREQUENCY DOMAIN

Step 2: Frequency Domain Perturbation

In the **Model Builder** window, under **Study 3 - Prestressed Frequency Domain** right-click **Step 2: Frequency Domain Perturbation** and choose **Copy**.

STUDY 5 - PRESTRESSED FREQUENCY DOMAIN WITH MANUFACTURING VARIATIONS

In the **Model Builder** window, right-click **Study 5 - Prestressed Frequency Domain with Manufacturing Variations** and choose **Paste Frequency-Domain Perturbation**.

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

Add an Evaluation Group to look at the changes in the amplitudes and sense capacitance. Reuse a previous Evaluation Group as a template but remove the sensitivity estimations (row 3 and 4 of the global evaluation table, because the formulas no longer apply after a large offset is introduced at zero rotation rate by the manufacturing variation.

RESULTS

Evaluation Group 2 - Study 3 - Prestressed Frequency Domain

In the **Model Builder** window, under **Results** right-click **Evaluation Group 2 - Study 3 - Prestressed Frequency Domain** and choose **Duplicate**.

Evaluation Group 4 - Study 5 - Prestressed Frequency Domain with Manufacturing Variations

1 In the **Model Builder** window, under **Results** click **Evaluation Group 2 - Study 3 - Prestressed Frequency Domain 1**.

2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group 4 - Study 5 - Prestressed Frequency Domain with Manufacturing Variations` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 5 - Prestressed Frequency Domain with Manufacturing Variations/Solution 12 (sol12)**.

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

Global Evaluation 1

1 In the **Model Builder** window, expand the **Evaluation Group 4 - Study 5 - Prestressed Frequency Domain with Manufacturing Variations** node, then click **Global Evaluation 1**.

2 In the Settings window for **Global Evaluation**, under the section **Expressions**, click on the 3rd row of the expression input table (with description "Sensitivity (aF/(deg/s))").

3 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

4 Click **Delete**.

5 Click **Delete**.

6 In the **Evaluation Group 4 - Study 5 - Prestressed Frequency Domain with Manufacturing Variations** toolbar, click **Evaluate**.

We see that there is a large imaginary part of the sense capacitance amplitude, even when there is no rotation. The reason is that the sidewall imperfection causes the drive mode to include a slight out-of-plane seesaw motion, which is exactly the same as the sense mode motion except for the 90 degrees phase difference.

The new drive mode shape can be visualized by slightly exaggerating the vertical deformation in the mode shape plot:

Mode Shape - Drive mode with sidewall

1 In the **Model Builder** window, under **Results** click **Mode Shape (solid) 1**.

2 In the **Settings** window for **3D Plot Group**, type `Mode Shape - Drive mode with sidewall` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (oe0 (nm),dt0 (nm),theta0 (deg))** list, choose **1: oe0=0 nm, dt0=0 nm, theta0=0.5 deg**.

4 In the **Model Builder** window, expand the **Mode Shape - Drive mode with sidewall** node.

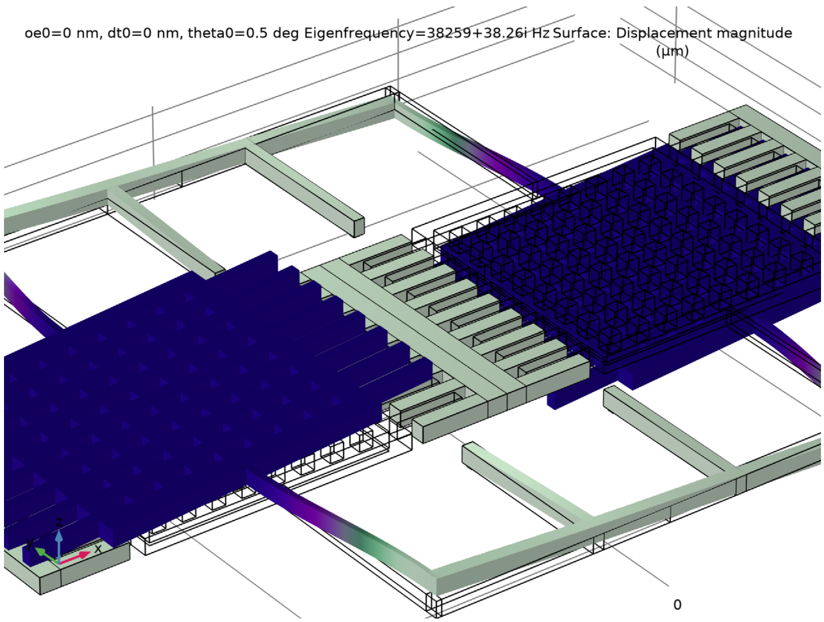
Deformation

1 In the **Model Builder** window, expand the **Results>Mode Shape - Drive mode with sidewall>Surface 1** node, then click **Deformation**.

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

3 In the **z-component** text field, type `w*20`.

4 In the **Mode Shape - Drive mode with sidewall** toolbar, click **Plot**.



This can be compared to the mode shape plot without sidewall from the original model, visualized using the same amount of exaggeration of the vertical deformation:

Mode Shape - Drive mode without sidewall

1 In the **Model Builder** window, under **Results** click **Mode Shape (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Mode Shape - Drive mode without sidewall` in the **Label** text field.

3 Locate the **Data** section. From the **Eigenfrequency (Hz)** list, choose **38262+38.262i**.

4 In the **Model Builder** window, expand the **Mode Shape - Drive mode without sidewall** node.

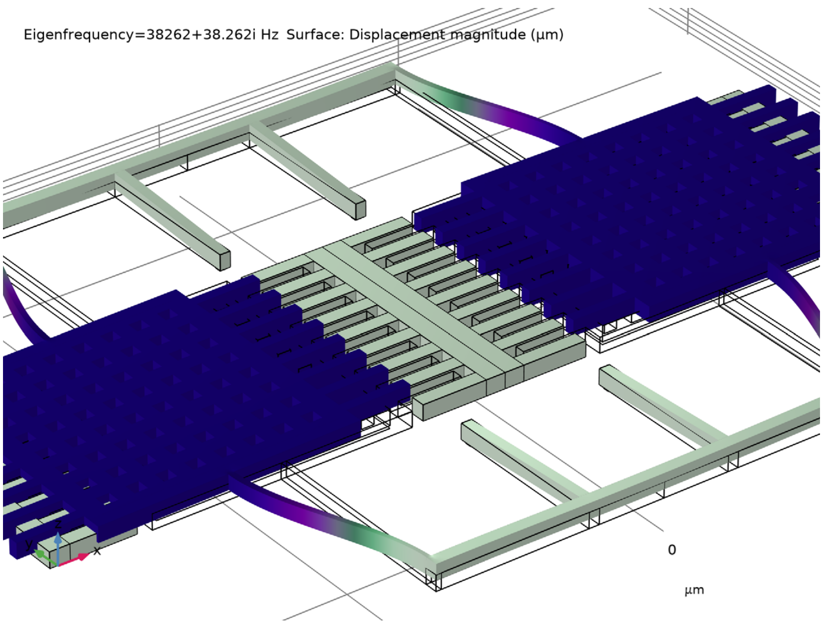
Deformation

1 In the **Model Builder** window, expand the **Results>Mode Shape - Drive mode without sidewall>Surface 1** node, then click **Deformation**.

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

3 In the **z-component** text field, type `w*20`.

4 In the **Mode Shape - Drive mode without sidewall** toolbar, click **Plot**.



There is no visible out-of-plane seesaw motion in this case. The slight common-mode out-of-plane motion does not contribute to the sense capacitance signal.

To visualize the real part of the sense mode amplitude, which contributes to the large offset of the real part of the sense capacitance amplitude in the case of sidewall imperfection, we start from the plot without sidewall:

Real Z displacement - No rotation, without sidewall

1 In the **Model Builder** window, under **Results** click **Real Z displacement - No rotation**.

2 In the **Settings** window for **3D Plot Group**, type `Real Z displacement - No rotation, without sidewall` in the **Label** text field.

3 In the **Model Builder** window, expand the **Real Z displacement - No rotation, without sidewall** node.

Volume 1

1 In the **Model Builder** window, expand the **Results>Real Z displacement - No rotation, without sidewall>Volume 1** node, then click **Volume 1**.

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

3 From the **Unit** list, choose **nm**.

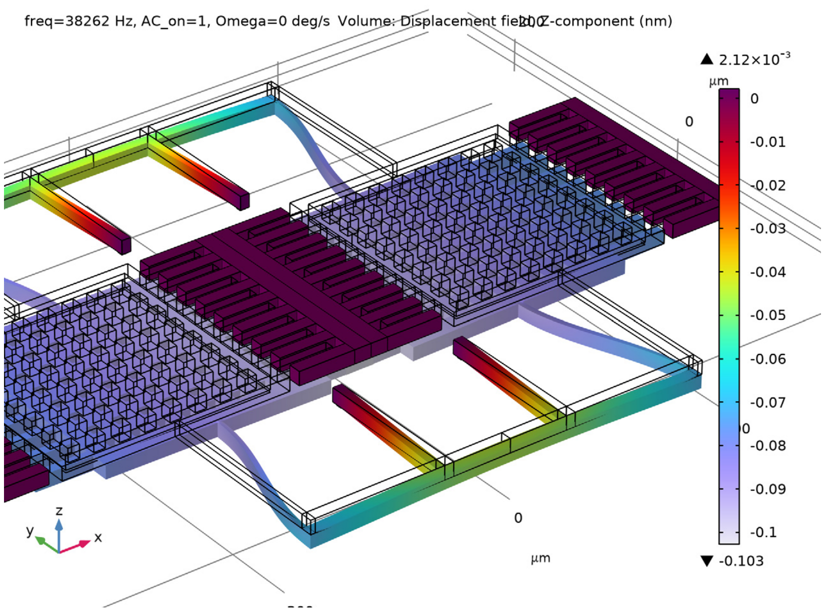
Deformation

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

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

3 In the **z-component** text field, type `w*20`.

4 In the **Real Z displacement - No rotation, without sidewall** toolbar, click **Plot**.



There is no visible out-of-plane seesaw motion in this case. The slight common-mode out-of-plane motion does not contribute to the sense capacitance signal.

Now duplicate the same plot for the case with sidewall imperfection:

Real Z displacement - No rotation, without sidewall

In the **Model Builder** window, under **Results** right-click **Real Z displacement - No rotation, without sidewall** and choose **Duplicate**.

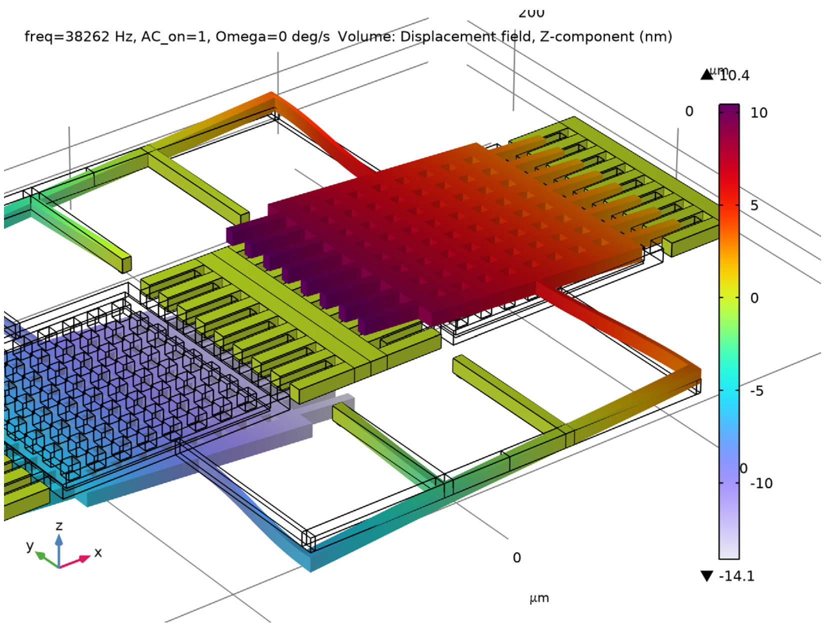
Real Z displacement - No rotation, with sidewall

1 In the **Model Builder** window, under **Results** click **Real Z displacement - No rotation, without sidewall 1**.

2 In the **Settings** window for **3D Plot Group**, type `Real Z displacement - No rotation, with sidewall` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 5 - Prestressed Frequency Domain with Manufacturing Variations/Solution 12 (sol12)**.

4 In the **Real Z displacement - No rotation, with sidewall** toolbar, click **Plot**.



There is a clear visible out-of-plane seesaw motion with an amplitude consistent with the global evaluation result.

Finally to visualize the shape change implemented by the **Deformed Geometry** interface, use a sidewall with zero azimuth and a large tilt angle of 10 degrees as an example. Create a stationary study with the solid mechanics physics disabled and the deformed geometry enabled. Use a Parametric Sweep to specify the sidewall angles.

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>Stationary**.
4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Solid Mechanics (solid)**.
5 Click **Add Study** in the window toolbar.
6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 6 - VISUALIZE SHAPE CHANGE

1 In the **Model Builder** window, click **Study 6**.
2 In the **Settings** window for **Study**, type `Study 6 - Visualize Shape Change` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.
2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3 Click **Add**.
4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
phi0 (Sidewall azimuth)	0	deg

5 Click **Add**.
6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta0 (Sidewall tilt)	10	deg

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

Create a plot to visualize the shape change. Use the node label as the plot title. Change the default black dataset edges from the default material frame to the geometry frame, to show the original undeformed geometry. Then use a line plot with uniform red color to show the device shape with the sidewall imperfection from the large tilt of 10 degrees.

RESULTS

Visualize shape change

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

2 In the **Settings** window for **3D Plot Group**, type `Visualize shape change` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 6 - Visualize Shape Change/ Solution 14 (sol14)**.

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

5 Locate the **Plot Settings** section. From the **Frame** list, choose **Geometry (Xg, Yg, Zg)**.

Line 1

1 Right-click **Visualize shape change** and choose **Line**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 In the **Visualize shape change** toolbar, click **Plot**.