Created in COMSOL Multiphysics 5.4

Piezoelectric Rate Gyroscope

Introduction

This model shows how to analyze a tuning fork based piezoelectric rate gyroscope. The reverse piezoelectric effect is used to drive an in-plane tuning fork mode. This mode is coupled to an out of plane mode by the Coriolis force and the resulting out of plane motion is sensed by the direct piezoelectric effect. The geometry of the tuning forks is designed so that the eigenfrequencies of the nearby modes are separated in frequency space. The frequency response of the system is computed and the rotation rate sensitivity is evaluated. Note that the model focuses on the performance of the sensor in a uniformly rotating reference frame. The model is based on the detailed analysis of a similar device presented in Ref. 1.

Model Definition

Figure 1 shows the geometry of the device indicating the key features.

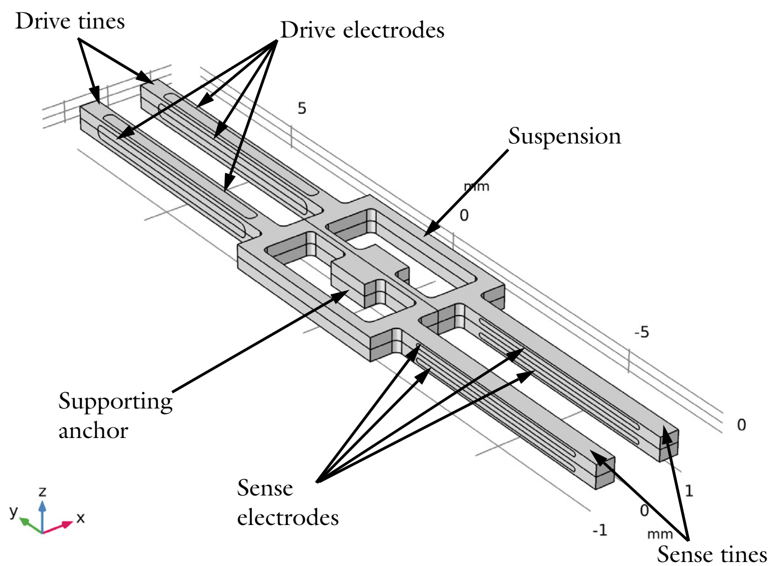


Figure 1: Device geometry showing the plane of symmetry through the center of the device and the key components of the gyroscope.

The packaging and fabrication of the device is discussed in Ref. 1. Here we provide a simple explanation of its principle of operation when operated in a rotating frame with no angular acceleration (Ref. 1 discusses the effects of an angular acceleration on the frequency response of the device in more detail). The gyroscope can be thought of as two tuning forks, coupled together by a suspension structure. The suspension is anchored to the package of the device which is in turn attached to the rotating object. The drive tines are driven close to their resonance in an in-plane mode, as shown in Figure 2. The sense tines are designed to have a resonance at a nearby, but distinct, frequency with a significant out of plane component to their motion, as shown in Figure 3. As the drive mode vibrates in the in-plane direction within the rotating frame a Coriolis body force acts on the structure which excites the out of plane sense mode. The Coriolis force ($\mathbf{F}_{\mathrm{cor}}$) is given by:

$$\mathbf{F}_{\mathrm{cor}} = -2\rho\Omega\times\frac{\partial\mathbf{u}}{\partial t}$$

Where ρ is the density of the material, Ω is the angular acceleration of the frame and **u** is the local velocity of the structure. From the above equation it is clear that the Coriolis force is maximal when the angular velocity of the frame is parallel to the long in-plane axis of the gyroscope structure. In this case the resulting force is in the out of plane direction and produces a corresponding out of plane motion of the drive tines. This motion causes reaction moments in the supporting suspension which in turn transfers these moments to the sense tines—driving the sense mode. Note that in this model the angular velocity vector is assumed to be parallel to the long axis of the device.

The tines are fabricated from single crystal quartz wafers with the crystallographic Z-axis aligned parallel to the normal of the wafer plane. The details of the design are discussed in Ref. 1, but the critical point is that the electrodes are patterned in such a way that both in-plane and in-phase out-of-plane motion of the sense tines is not detected by the sense electrodes. This leads to the rejection of unwanted signals in the output of the sensor.

In general, for resonant structures like this model, a very fine mesh is required to achieve accurate frequency response results. In the interest of saving time, we choose to use a relatively coarse mesh for this tutorial. As a result the resonant peak will shift if a more refined mesh is used instead.

Results and Discussion

Figure 2 shows the eigenmode corresponding to the drive mode and Figure 3 shows that corresponding to the sense mode. Both the in-plane and out-of-plane motions of these modes are shown separately in the figures.

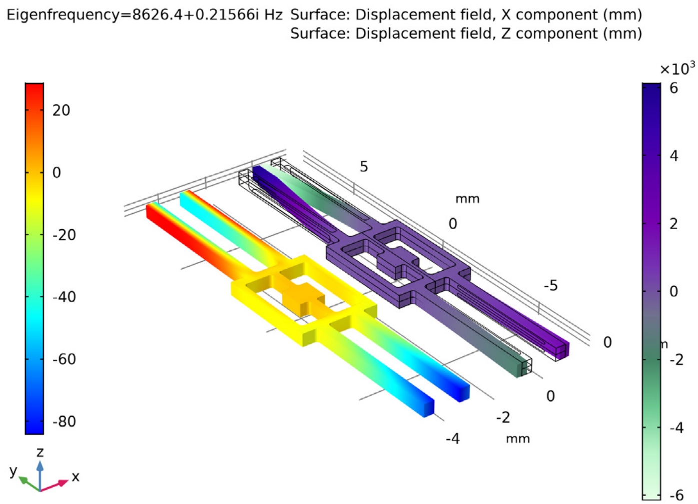


Figure 2: Drive mode, showing both in-plane motion (right) and out-of-plane motion (left). Note that the amplitude scale is arbitrary—only the relative value of the in plane and out of plane displacements has physical significance.

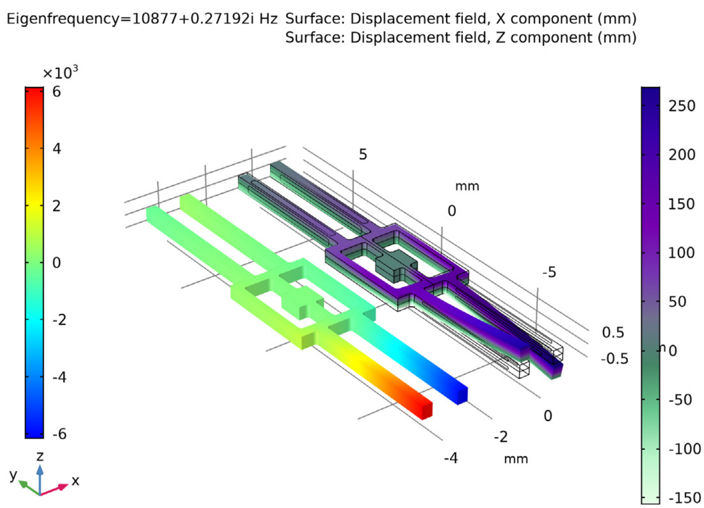


Figure 3: Sense mode, showing out-of-plane motion (left) and in-plane motion (right). Note that the amplitude scale is arbitrary—only the relative value of the in plane and out of plane displacements has physical significance.

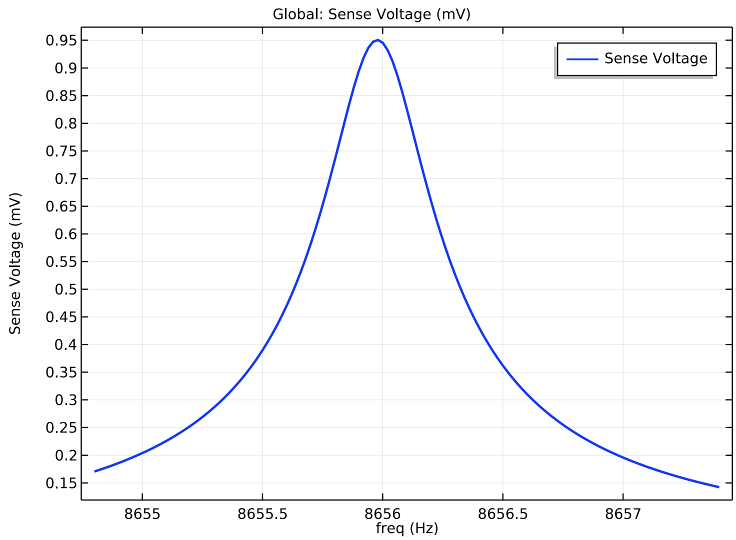


Figure 4: Sense voltage vs. drive frequency with an applied sinusoidal drive voltage of amplitude 2 V and an angular acceleration of 64 deg/s.

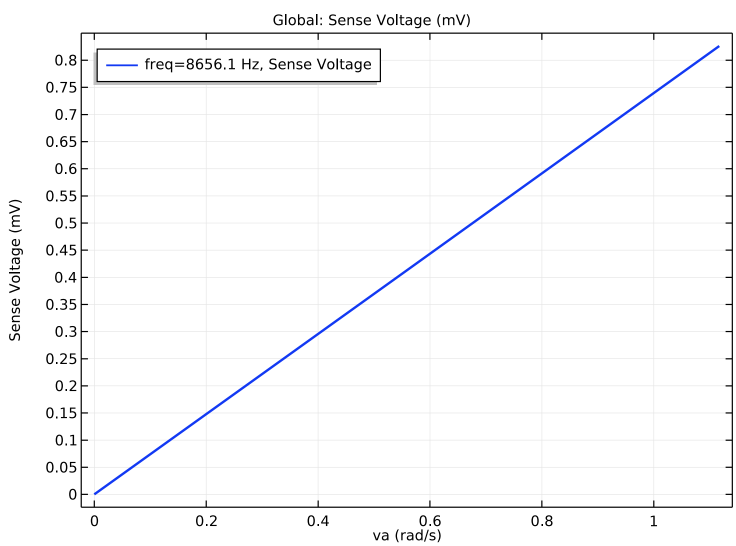


Figure 5: Sense voltage vs. angular acceleration at a drive voltage amplitude of 2 V and a frequency of 8656 Hz.

Figure 4 shows the response of the device as the frequency of the drive voltage waveform is varied. A clear peak in the response close to the drive frequency, at approximately 8656 Hz is apparent. This is the optimum drive frequency for the device. Figure 5 shows the sense voltage against the angular acceleration with a 2 V drive voltage at a frequency close to this optimum. As expected the response of the sensor is linear, with a sensitivity of approximately 0.015 mV /(deg/s).

Reference

1. S.D. Senturia, "A Piezoelectric Rate Gyroscope," *Microsystem Design*, chapter 21, Springer, 2000.

Application Library path: `MEMS_Module/Piezoelectric_Devices/piezoelectric_rate_gyroscope`

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 **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Piezoelectric Devices**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

Add some global parameters.

GLOBAL DEFINITIONS

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
va	64[deg/s]	1.117 rad/s	Rotation angular velocity

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**.

1 In the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `piezoelectric_rate_gyroscope_geom_sequence.mph`.

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

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

The **Adaptive Frequency Sweep** study step will generate a high resolution frequency sweep. To avoid large file size, create an "explicit selection" to store solution data only on the external surfaces of the modeling domain.

DEFINITIONS

Explicit 1

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

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

3 Select the **All domains** check box.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

5 In the **Label** text field, type `External surfaces`.

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>Quartz LH (1978 IEEE)**.

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

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

SOLID MECHANICS (SOLID)

Mechanical Damping 1

1. In the **Physics** toolbar, click **Attributes** and choose **Mechanical Damping**.
2. In the **Settings** window for **Mechanical Damping**, locate the **Damping Settings** section.
3. From the **Damping type** list, choose **Isotropic loss factor**.
4. From the η_s list, choose **User defined**. In the associated text field, type `5e-5`.

Fixed Constraint 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
2. Select Boundaries 56 and 80 only.

MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.
3. From the **Sequence type** list, choose **User-controlled mesh**.

Size

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Finer**.

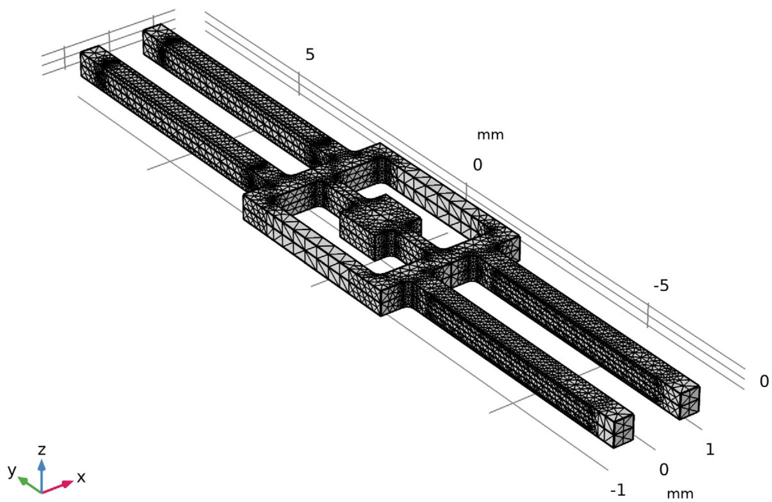
Free Tetrahedral 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.
2. In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domain 1 only.

Copy Domain 1

1. In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Copy Domain**.
2. Select Domain 1 only.
3. In the **Settings** window for **Copy Domain**, locate the **Destination Domains** section.
4. Select the **Active** toggle button.
5. Select Domains 2–4 only.

6 In the **Home** toolbar, click **Build Mesh**.



STUDY 1

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

2 In the **Settings** window for **Study**, type `Mechanical Eigenmode` in the **Label** text field.

MECHANICAL EIGENMODE

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Mechanical Eigenmode** 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 `10`.

5 Select the **Search for eigenfrequencies around** check box.

6 In the associated text field, type `1e3`.

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

8 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Electrostatics (es)** interface.

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

RESULTS

Mode Shape (solid)

The default mode shape plot shows a surface plot of the total displacement. To provide a deeper insight into the mode shapes, let us plot instead the x and z displacements in two separate surface plots.

Change the default surface plot to plot the x displacement.

Surface 1

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

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

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

Duplicate the surface plot to plot the z displacement.

Surface 2

1 Right-click **Results>Mode Shape (solid)>Surface 1** and choose **Duplicate**.

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

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

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Deformation

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Deformation**.

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

3 In the **x component** text field, type `-w_f*1.3`.

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

5 In the **z component** text field, type `0`.

6 Locate the **Scale** section. Select the **Scale factor** check box.

7 In the associated text field, type `1`.

Mode Shape (solid)

Turn on color legend to see the relative amplitude of the x and z displacements.

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

2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

3 Select the **Show legends** check box.

4 From the **Position** list, choose **Alternating**.

Plot the mode shape of the drive mode.

5 Locate the **Data** section. From the **Eigenfrequency (Hz)** list, choose **8626.4+0.21566i**.

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

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

Plot the sense mode, which is at a higher frequency.

8 From the **Eigenfrequency (Hz)** list, choose **10877+0.27192i**.

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

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

SOLID MECHANICS (SOLID)

Add rotating frame physics.

Rotating Frame 1

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

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

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

4 Locate the **Rotating Frame** section. From the **Axis of rotation** list, choose **y-axis**.

5 In the Ω text field, type `va`.

6 Select the **Coriolis force** check box.

ELECTROSTATICS (ES)

Add boundary conditions for the drive and sense electrodes.

1 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 Boundaries 36, 37, 117, 118, 133, and 134 only.

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

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

5 In the **Label** text field, type `Drive Terminal 1`.

Terminal 2

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

2 Select Boundaries 32, 33, 54, 55, 121, and 122 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 `-1`.

6 In the **Label** text field, type `Drive Terminal 2`.

Terminal 3

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

2 Select Boundaries 29, 40, 111, and 126 only.

3 In the **Settings** window for **Terminal**, type `Sense Terminal 1` in the **Label** text field.

Terminal 4

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

2 Select Boundaries 28, 41, 112, and 125 only.

3 In the **Settings** window for **Terminal**, type `Sense Terminal 2` in the **Label** text field.

Set up and solve an **Adaptive Frequency Sweep** study, which is optimized for resolving narrow resonant peaks without excessive computation.

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 2

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

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Adaptive Frequency Sweep

In the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>
Adaptive Frequency Sweep**.

FREQUENCY RESPONSE

Step 1: Adaptive Frequency Sweep

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

2 In the **Frequencies** text field, type `range(8654.8,0.02,8657.4)`.

3 From the **AWE expression type** list, choose **User controlled**.

4 In the table, enter the following settings:

Asymptotic waveform evaluation (AWE) expressions
abs(comp1.es.V0_4-comp1.es.V0_3)/1e-3

To reduce file size, only store solution data on the external surfaces.

5 Locate the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

6 Under **Selections**, click **Add**.

7 In the **Add** dialog box, select **External surfaces** in the **Selections** list.

8 Click **OK**.

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

RESULTS

Mode Shape (solid) 1

1 In the **Model Builder** window, under **Results** right-click **Mode Shape (solid)** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Frequency Response: Displacement` in the **Label** text field.

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

4 In the **Frequency Response: Displacement** toolbar, click **Plot**.

1D Plot Group 3

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

2 In the **Settings** window for **1D Plot Group**, type `Frequency Response: Sense Voltage` in the **Label** text field.

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

Global 1

1 Right-click **Frequency Response: Sense Voltage** and choose **Global**.

2 In the **Settings** window for **Global**, type `Sense Voltage` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
abs(es.V0_4-es.V0_3)	mV	Sense Voltage

4 Click to expand the **Coloring and Style** section. In the **Width** text field, type 2.

5 In the **Frequency Response: Sense Voltage** toolbar, click **Plot**.

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 3

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `8656.1`.

3 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
va	0 32 64	deg/s

6 In the **Model Builder** window, click **Study 3**.

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

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

9 In the **Label** text field, type `Sensitivity`.

For this study, the default MUMPS linear solver does not perform well, so we use the PARDISO solver instead.

To change the linear solver, first generate the solver sequence.

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

SENSITIVITY

Solution 3 (sol3)

1 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

Change the linear solver from MUMPS to PARDISO.

2 In the **Model Builder** window, expand the **Sensitivity>Solver Configurations> Solution 3 (sol3)>Stationary Solver 1** node, then click **Direct**.

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

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

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

RESULTS

Frequency Response: Sense Voltage 1

1 In the **Model Builder** window, under **Results** right-click **Frequency Response: Sense Voltage** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Sensitivity/Solution 3 (sol3)**.

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

5 In the **Label** text field, type `Sensitivity: Sense Voltage vs. Angular Velocity`.

Sense Voltage

1 In the **Model Builder** window, expand the **Results> Sensitivity: Sense Voltage vs. Angular Velocity** node, then click **Sense Voltage**.

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

3 From the **Unit** list, choose **rad/s**.

4 In the **Sensitivity: Sense Voltage vs. Angular Velocity** toolbar, click **Plot**.