



Model created in COMSOL Multiphysics 6.4

# Piezoelectric Energy Harvester with Uncertainty Quantification

## *Introduction*

---

This tutorial demonstrates how to use the Uncertainty Quantification Module to analyze the performance variation of a piezoelectric energy harvester. The piezoelectric energy harvester is a “seismic” energy harvester designed to convert vibrational energy to electric energy when mounted to a vibrating piece of machinery. This tutorial studies how variations in manufacturing processes translate to variation in the device’s output power as the quantity of interest (QoI). Different **Uncertainty Quantification** (UQ) studies are deployed to provide design-of-experiment data from FEM simulations and to generate a surrogate model for evaluating the QoI. The design objective is the output power should not fall below 1 mW in its normal operating range.

## *Model Definition*

---

The energy harvester analyzed in this model consists of a mass suspended by a piezoelectric bimorph with one end clamped to a vibrating machinery as shown in [Figure 1](#). The geometry is a simplified version of the device featured in the model [Piezoelectric Energy Harvester](#) and has been parameterized for the uncertainty quantification studies. The bimorph has two output electrodes embedded within it and two ground electrodes on the exterior surfaces of the cantilever beam. This configuration ensures that same voltage is induced on the output electrodes, even though the stress above and below the neutral layer is of opposite sign. Electric power is generated when the harvester is subjected to vibrations that cause the proof mass to swing and the piezoelectric layers comprising the beam to bend. The device is analyzed in a vibrating reference frame by the application of a sinusoidal body load. This tutorial begins with the setup of the reference **Frequency Domain** study assuming the vibrating machinery has a frequency range of 74-82 Hz.

Manufacturing variations produce device-to-device performance variation. For example, variations in thickness of the piezoelectric layers, or `t_piezo`, results in variation in device output power. In this context, `t_piezo` is an input parameter and output power is the quantity of interest (QoI). Other input parameters are `t_mass` and `R_load`. Here, we consider the case where the input parameters are uncorrelated. The [Modeling Instructions](#) show how to specify the distribution function associated with each input parameter. The subsequent **Uncertainty Quantification** (UQ) studies lead to the probability distributions for the output power. UQ studies are added in sequence, starting with **Screening**, **MOAT** and ending with a **Reliability Analysis** with a reliability criteria involving output power. We also assume the energy harvester is intended to be connected to a band-pass filter with frequency range of 77.5-78.5 Hz (not included in the model) so the **Reliability Analysis** will apply the criteria to this narrower frequency range.

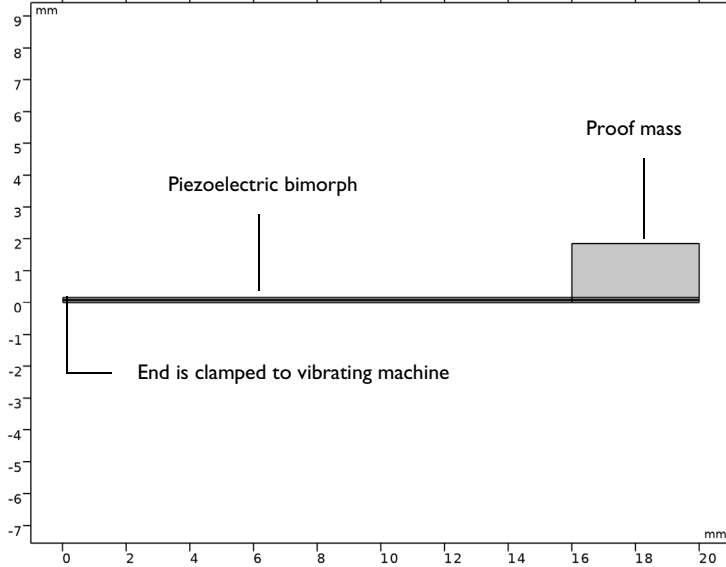


Figure 1: 2D model geometry, showing the major components of the energy harvester, including the piezoelectric bimorph and proof mass.

## THE UNCERTAINTY QUANTIFICATION STUDIES

The Uncertainty Quantification Module provides five different study types:

- **Screening, MOAT**
  - Identifies the most influential inputs, for each QoI
  - Is based on the Morris One-At-a-Time (MOAT) method
  - Outputs MOAT mean and MOAT standard deviation values
- **Sensitivity Analysis**
  - Computes the fraction of impact for the inputs, for each QoI
  - Outputs first-order and total Sobol indices
- **Uncertainty Propagation**
  - Computes the statistical variation of the QoI
  - Outputs a kernel density estimation (KDE) plot representing an estimate of the probability distribution of the QoI

- **Reliability Analysis**
  - Computes the probability for the fulfillment of a condition based on the QoI
  - For example, what is the probability  $P(\phi > 0.1)$ ?
- **Inverse Uncertainty Quantification**
  - Uses experimental data to backtrack and uncover the statistical properties of certain input parameters, starting with some initial assumptions about them (prior probability distribution).
  - Applies surrogate models and statistical methods to refine the characterization of these inputs, providing (posterior) probability distributions, as well as a confidence interval table containing the mean, standard deviation, min/max values, and confidence bounds.

This example illustrates the first four of these study types. For more information, see the *Uncertainty Quantification Module User's Guide*.

## **SURROGATE MODELS**

To get statistical data based on a physics model you need to run a lot of simulations, varying the parameters of the inputs according to their probability distributions. For a 3D model, this might be computationally unfeasible. To get around this problem, the Uncertainty Quantification Module first builds up a so-called surrogate model that is used for sensitivity analysis, uncertainty propagation, and reliability analysis (but not for screening).

This process is typically adaptive and the surrogate model can approximate the original model to a high degree of accuracy (which can be modified by the user). The Uncertainty Quantification Module uses two different types of surrogate models:

- Sparse Polynomial Chaos Expansion (SPCE)
  - This surrogate model improves its accuracy by adaptively solving the full model and thereby adding new QoI data by using sequential Latin hypercube sampling.
- Gaussian Process (GP)
  - This surrogate model improves its accuracy by, using information from the current Gaussian Process surrogate model, adaptively solve the full model for new carefully selected sets of parameter values.

In this tutorial, a set of UQ studies are done with the assumption that the input parameters are not correlated.

## Results and Discussion

Figure 2 shows the input mechanical power and the power harvested (in mW) as well as the peak voltage induced across the piezoelectric bimorph (in V) as a function of frequency when the energy harvester is excited by a sinusoidal acceleration. The electrical load is 12 k $\Omega$ . The response of the system shows a peak at 78 Hz.

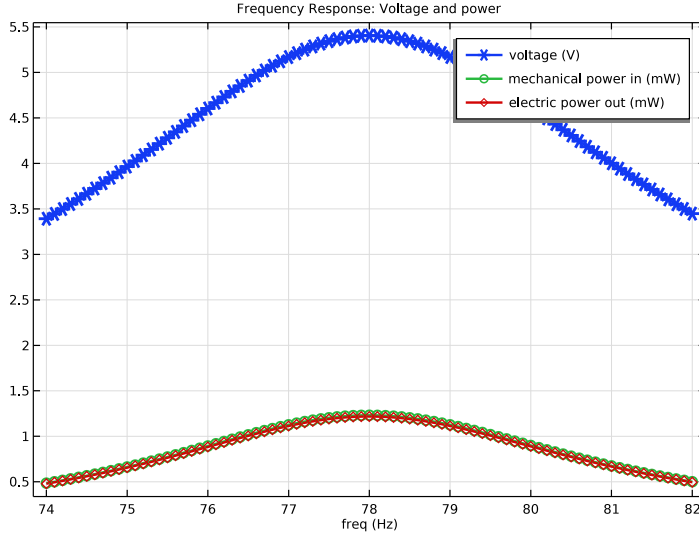


Figure 2: Energy harvester input mechanical power and the power harvested (in mW) as well as the peak voltage induced across the piezoelectric bimorph (in V) versus excitation frequency. The load impedance is 12 k $\Omega$ .

Results from the screening study are shown in Figure 3. They indicate that the piezoelectric layer thickness  $t_{\text{piezo}}$  and mass thickness  $t_{\text{mass}}$  parameters are the most influential on the quantity of interest. On the other hand, the load resistance  $R_{\text{load}}$  is not influential so this parameter is excluded from subsequent analyses.

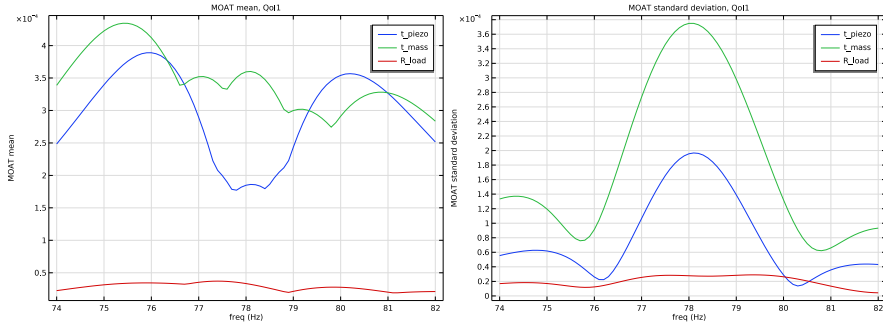


Figure 3: MOAT mean (left) and standard deviation (right) obtained from the Screening study.

Figure 4 shows the results of the sensitivity analysis: the first-order and the total Sobol indices, or the sensitivity of the quantity of interest to the input parameters. Outside of the region between 77 and 79 Hz, the values oscillate because a tolerance value of 0.01 was chosen to save time. Using the default tolerance value of 0.001 will result in smoother plots.

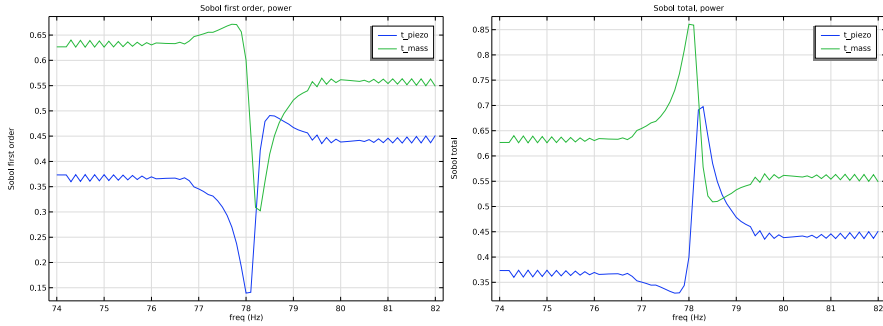


Figure 4: The first order (left) and the total (right) Sobol indices, or the sensitivity of the quantity of interest to the input parameters, from the Sensitivity Analysis study.

Figure 5 shows the mean of the quantity of interest (QoI) due to the variations in the input parameters with 95% prediction interval resulting from **Uncertainty Propagation** study.

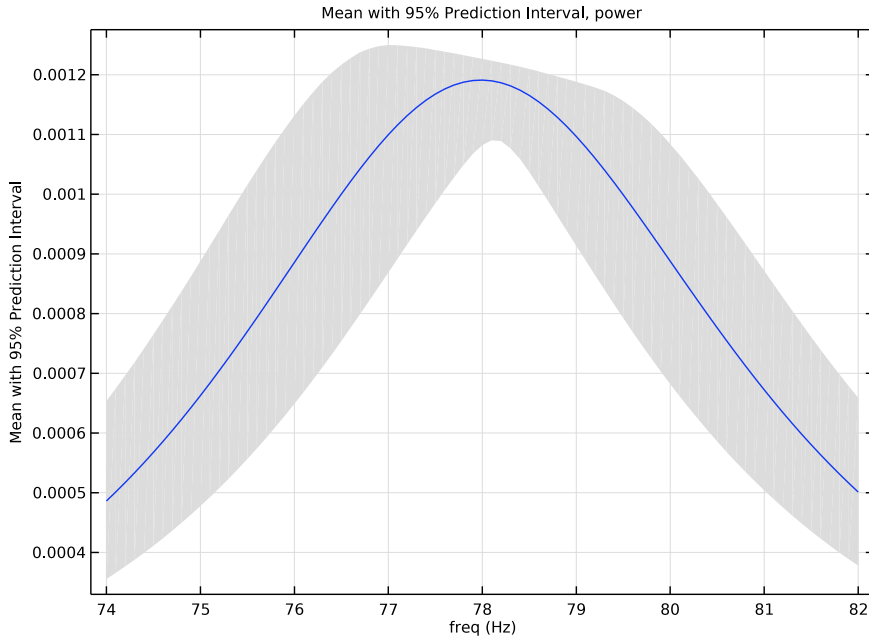


Figure 5: The mean of the quantity of interest due to the variations in the input parameters with 95% prediction interval from the Uncertainty Propagation study.

## Reference

I. E. Lefeuvre, D. Audiger, C. Richard, and D. Guyomar, “Buck-Boost Converter for Sensorless Power Optimization of Piezoelectric Energy Harvester,” *IEEE Transactions on Power Electronics*, vol. 22, no. 5, pp. 2018–2025, 2007.


**Application Library path:** MEMS\_Module/Piezoelectric\_Devices/  
piezoelectric\_energy\_harvester\_uncertainty\_quantification

## Modeling Instructions




Start by creating a new 2D model with a **Piezoelectricity** multiphysics interface and then add a **Frequency Domain** study.

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 **AC/DC** > **Electromagnetics and Mechanics** > **Piezoelectricity** > **Piezoelectricity, Solid**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies** > **Frequency Domain**.
- 6 Click  **Done**.

## GEOMETRY I

Use millimeters as the geometry unit.

- 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**.

Define and enter the values for the following parameters.

## 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
acc	1	1	Acceleration
R_load	12[kohm]	12000 $\Omega$	Load resistance
w_plate	14[mm]	0.014 m	Out-of-plane dimension
t_piezo	0.04[mm]	4E-5 m	Piezoelectric layer thickness
w_mass	4[mm]	0.004 m	Mass width
t_mass	1.7[mm]	0.0017 m	Mass thickness

Create the geometry for the energy harvester using some of the parameters defined previously.


## GEOMETRY I

### 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 20.
- 4 In the **Height** text field, type  $0.12+t\_piezo$ .
- 5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	0.06
Layer 2	$t\_piezo$

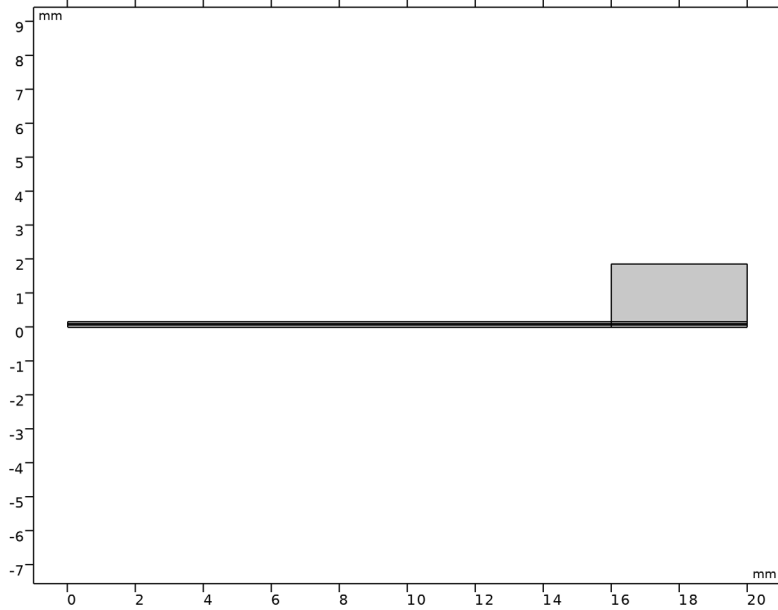
### Rectangle 2 (r2)

- 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  $w\_mass$ .
- 4 In the **Height** text field, type  $t\_mass$ .
- 5 Locate the **Position** section. In the **x** text field, type  $20-w\_mass$ .
- 6 In the **y** text field, type  $0.12+t\_piezo$ .

### Partition Domains 1 (pard1)



- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Partition Domains**.
- 2 On the object **r1**, select Domains 1–3 only.
- 3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.
- 4 Click to select the  **Activate Selection** toggle button for **Vertices defining line segments**.
- 5 From the **Partition with** list, choose **Extended edges**.
- 6 On the object **r2**, select Boundary 4 only.
- 7 In the **Geometry** toolbar, click  **Build All**.

8 In the **Model Builder** window, click **Geometry 1**.



Add materials and assign the domains they belong to.

#### ADD MATERIAL

- 1 In the **Materials** toolbar, click  **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.
- 3 In the tree, select **Piezoelectric** > **Lead Zirconate Titanate (PZT-5A)**.
- 4 Click the **Add to Component** button in the window toolbar.
- 5 In the tree, select **Built-in** > **Structural steel**.
- 6 Click the **Add to Component** button in the window toolbar.
- 7 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

#### MATERIALS

*Structural steel (mat2)*



Select Domains 2, 5, and 7 only.

Specify the settings for the **Electrostatics** interface.



## ELECTROSTATICS (ES)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electrostatics (es)**.
- 2 In the **Settings** window for **Electrostatics**, locate the **Thickness** section.
- 3 In the  $d$  text field, type `w_plate`.



### *Charge Conservation, Piezoelectric 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** > **Electrostatics (es)** click **Charge Conservation, Piezoelectric 1**.
- 2 In the **Settings** window for **Charge Conservation, Piezoelectric**, locate the **Domain Selection** section.
- 3 Click  **Clear Selection**.
- 4 Click  **Paste Selection**.
- 5 In the **Paste Selection** dialog, type 1 3 4 6 in the **Selection** text field.
- 6 Click **OK**.

### *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 Click  **Paste Selection**.
- 4 In the **Paste Selection** dialog, type 2 7 9 15 in the **Selection** text field.
- 5 Click **OK**.

### *Boundary Terminal 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Boundary Terminal**.
- 2 In the **Settings** window for **Boundary Terminal**, locate the **Boundary Selection** section.
- 3 Click  **Paste Selection**.
- 4 In the **Paste Selection** dialog, type 4 6 11 13 in the **Selection** text field.
- 5 Click **OK**.
- 6 In the **Settings** window for **Boundary Terminal**, locate the **Terminal** section.
- 7 From the **Terminal type** list, choose **Circuit**.


Specify the settings for the **Solid Mechanics** interface.

## SOLID MECHANICS (SOLID)



### *Linear Elastic Material I*

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


### *Damping I*

- 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**.
- 4 From the  $\eta_s$  list, choose **User defined**. In the associated text field, type 0.001.


### *Piezoelectric Material I*

- 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 Click  **Clear Selection**.
- 4 Click  **Paste Selection**.
- 5 In the **Paste Selection** dialog, type 1 3 4 6 in the **Selection** text field.
- 6 Click **OK**.

### *Mechanical Damping I*


- 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  $\eta_s$  list, choose **User defined**. In the associated text field, type 0.001.


### *Body Load I*

- 1 In the **Physics** toolbar, click  **Domains** and choose **Body Load**.
- 2 In the **Settings** window for **Body Load**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **All domains**.
- 4 Locate the **Force** section. Specify the  $\mathbf{f}_V$  vector as

-solid.rho\*g\_const\*acc | y



### *Fixed Constraint I*

- 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 Click  **Paste Selection**.
- 4 In the **Paste Selection** dialog, type 1 3 5 in the **Selection** text field.
- 5 Click **OK**.


Add and specify the settings for the **Electrical Circuit** interface.

### ADD PHYSICS

- 1 In the **Physics** toolbar, click  **Add Physics** to open the **Add Physics** window.
- 2 Go to the **Add Physics** window.
- 3 In the tree, select **AC/DC** > **Electrical Circuit (cir)**.
- 4 Click the **Add to Component 1** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Physics** to close the **Add Physics** window.

### ELECTRICAL CIRCUIT (CIR)


#### Resistor 1 (R1)

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

Label	Node names
n	0

- 4 Locate the **Device Parameters** section. In the  $R$  text field, type  $R_{load}$ .


#### External I-Terminal 1 (term1)


- 1 In the **Electrical Circuit** toolbar, click  **External I-Terminal**.
- 2 In the **Settings** window for **External I-Terminal**, locate the **Node Connections** section.
- 3 In the **Node name** text field, type 1.
- 4 Locate the **External Terminal** section. From the  $V$  list, choose **Terminal voltage (es/term1)**.

Create a structured mesh.

### MESH 1

#### Edge 1



- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Edge**.
- 2 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.

- 3 Click  **Paste Selection**.
- 4 In the **Paste Selection** dialog, type 1 3 5 in the **Selection** text field.
- 5 Click **OK**.


#### *Distribution 1*

- 1 Right-click **Edge 1** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3 In the **Number of elements** text field, type 3.


#### *Edge 2*

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Edge**.
- 2 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.
- 3 Click  **Paste Selection**.
- 4 In the **Paste Selection** dialog, type 7 15 in the **Selection** text field.
- 5 Click **OK**.



#### *Distribution 1*

- 1 Right-click **Edge 2** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.
- 3 In the list box, select **15**.
- 4 Click  **Remove from Selection**.
- 5 Select Boundary 7 only.
- 6 Locate the **Distribution** section. In the **Number of elements** text field, type 160.

#### *Distribution 2*

- 1 In the **Model Builder** window, right-click **Edge 2** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.
- 3 In the list box, select **7**.
- 4 Click  **Remove from Selection**.
- 5 Select Boundary 15 only.
- 6 Locate the **Distribution** section. In the **Number of elements** text field, type 40.

#### *Edge 3*

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Edge**.
- 2 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.
- 3 Click  **Paste Selection**.


4 In the **Paste Selection** dialog, type 14 in the **Selection** text field.

5 Click **OK**.

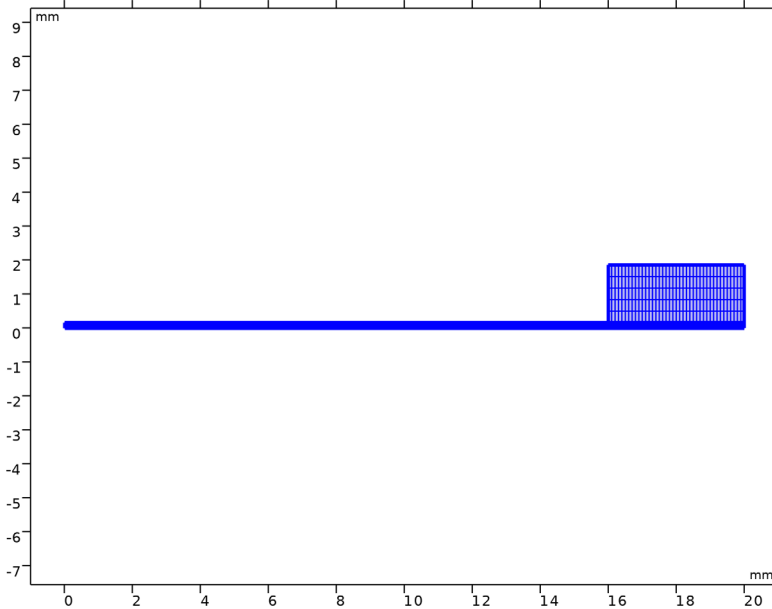
#### *Distribution 1*

Right-click **Edge 3** and choose **Distribution**.

#### *Mapped 1*

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

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



### **DEFINITIONS**

Define a nonlocal integration coupling to calculate mechanical power input later when plotting results.

#### *Integration 1 (intop1)*

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

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.


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

Set up the **Frequency Domain** study.

## STUDY 1

- 1 In the **Model Builder** window, click **Study 1**.
- 2 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 3 Clear the **Generate default plots** checkbox.


### *Step 1: Frequency Domain*

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.
- 2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
- 3 In the **Frequencies** text field, type range (74, 0.1, 82).
- 4 In the **Study** toolbar, click  **Compute**.

Create a plot of electric power output.

## RESULTS

### *Frequency Response: Voltage and Power*

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Frequency Response: Voltage and Power in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type Frequency Response: Voltage and power.

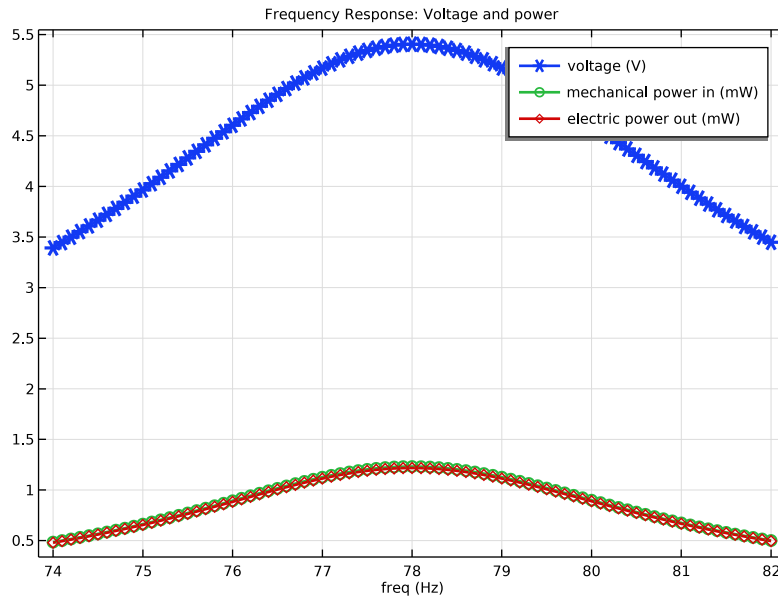
### *Global 1*

- 1 Right-click **Frequency Response: Voltage and Power** 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
<code>abs(cir.R1.v)</code>	V	voltage (V)
<code>0.5*intop1(realdot(-solid.rho*g_const*acc,solid.u_tY))*w_plate</code>	mW	mechanical power in (mW)
<code>0.5*realdot(cir.R1.i,cir.R1.v)</code>	mW	electric power out (mW)

- 4 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.
- 5 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

6 In the **Frequency Response: Voltage and Power** toolbar, click  **Plot**.



Next, add a screening analysis to see which input parameters are most significantly impacting the resonance frequency (QoI). The screening study is added as a study reference which means it refers back to the already defined **Frequency Domain** study. The parameters that participate in the uncertainty quantification are all assumed to be normally distributed around their nominal values, according to the instructions below. The mean and standard deviation, as well as the max and min limits are all defined in terms of their nominal parameters (from **Global Definitions > Parameters**).

#### STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **More Study Extensions > Add Uncertainty Quantification Study Using Study Reference**.

#### STUDY 2

##### *Uncertainty Quantification*

1 In the **Settings** window for **Uncertainty Quantification**, locate the **Quantities of Interest** section.

2 Click  **Add**.

3 In the table, enter the following settings:

Name	Expression	Include study-dependent input
	$0.5 * \text{realdot}(\text{cir.R1.i}, \text{cir.R1.v})$	Configure study-dependent input

4 Locate the **Input Parameters** section. Click **+ Add**.

5 In the table, enter the following settings:

Parameter	Source type	Source description
t_piezo (Piezoelectric layer thickness)	Analytic	Normal

6 From the **Distribution** list, choose **Normal( $\mu, \sigma$ )**.

7 In the **Mean** text field, type t\_piezo.

8 In the **Standard deviation** text field, type t\_piezo\*0.01.

9 Click **+ Add**.

10 In the table, enter the following settings:

Parameter	Source type	Source description
t_mass (Mass thickness)	Analytic	Normal

11 From the **Distribution** list, choose **Normal( $\mu, \sigma$ )**.

12 In the **Mean** text field, type t\_mass.

13 In the **Standard deviation** text field, type t\_mass\*0.01.

14 Click **+ Add**.

15 In the table, enter the following settings:

Parameter	Source type	Source description
R_load (Load resistance)	Analytic	Normal


16 From the **Distribution** list, choose **Normal( $\mu, \sigma$ )**.

17 In the **Mean** text field, type R\_load.

18 In the **Standard deviation** text field, type R\_load\*0.01.

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

20 In the **Settings** window for **Study**, type Study 2: Screening in the **Label** text field.

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

Next, add a **Sensitivity Analysis** study. Use the results from the screening to decide which parameters to include in the sensitivity analysis. Sensitivity is more computationally demanding than screening and for this reason we would prefer to pick a subset of the parameters used for the screening study. In this example, we exclude `R_load`. We do not need to type all of the uncertainty quantification parameters again but we can define the new Uncertainty Quantification study for the sensitivity analysis by reusing the information in the screening study.

**22** In the **Model Builder** window, right-click **Uncertainty Quantification** and choose **Add New Uncertainty Quantification Study For > Sensitivity Analysis**.

### STUDY 3: SENSITIVITY ANALYSIS

Delete the input parameter `R_load` because of its small effect on QoI. To minimize computation time, increase tolerance to 0.01. In Sampling Settings, set the maximum number of input points to 40 and the initial number of input points to 20.

**1** In the **Model Builder** window, under **Study 3: Sensitivity Analysis** click **Uncertainty Quantification**.

**2** In the **Settings** window for **Uncertainty Quantification**, locate the **Uncertainty Quantification Settings** section.

**3** Find the **Surrogate model settings** subsection. In the **Relative tolerance** text field, type 0.01.

**4** Locate the **Quantities of Interest** section. In the table, enter the following settings:

Name	Expression	Include study-dependent input
power	<code>0.5*realdot(cir.R1.i, cir.R1.v)</code>	Configure study-dependent input


**5** Locate the **Input Parameters** section. In the table, click to select the cell at row number 3 and column number 1.

**6** Click  **Delete**.

**7** Locate the **Sampling Settings** section. From the **Number of input points type** list, choose **Manual**.

**8** In the **Maximum number of input points** text field, type 40.

**9** In the **Initial number of input points** text field, type 20.

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


The sensitivity analysis is based on the Sobol method, also known as variance-based sensitivity analysis. The result of the sensitivity analysis is a set of Sobol indices and an associated Sobol table and Sobol plot. There are two different types of Sobol indices: first order-index and total index. The first-order index of a parameter shows the sensitivity by varying this parameter alone. The total index shows how much a parameter contributes to the overall sensitivity. In this case, the first and total indices are equal, up to the computed accuracy, for all parameters which indicates very little or no interaction between the parameters. The Sobol plot indicates that piezoelectric layer thickness and the mass layer thickness are most sensitive. This is consistent with the screening results.

Next, add an **Uncertainty Propagation** study. For this study we will keep all the parameters to get an accurate estimate of the uncertainties

11 In the **Model Builder** window, right-click **Uncertainty Quantification** and choose **Add New Uncertainty Quantification Study For > Uncertainty Propagation**.

#### STUDY 4: UNCERTAINTY PROPAGATION

##### *Uncertainty Quantification 3*

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

The uncertainty propagation study computes a so-called kernel density estimation or KDE. You can think of the KDE as a smooth form of a histogram showing an estimate of the probability density function of the quantity of interest, given the input parameters and their distributions.

Finally, add a **Reliability Analysis** study.

##### *Uncertainty Quantification*

In the **Model Builder** window, under **Study 4: Uncertainty Propagation** right-click **Uncertainty Quantification** and choose **Add New Uncertainty Quantification Study For > Reliability Analysis**.

#### STUDY 5: RELIABILITY ANALYSIS, EGRA

1 In the **Model Builder** window, under **Study 5: Reliability Analysis, EGRA** click **Uncertainty Quantification**.

2 In the **Settings** window for **Uncertainty Quantification**, locate the **Uncertainty Quantification Settings** section.

3 From the **Compute action** list, choose **Compute and analyze**.


4 Locate the **Quantities of Interest** section. In the table, enter the following settings:

Name	Expression	Include study-dependent input	True if	Threshold
power	0.5*realdot(cir.R1.i, cir.R1.v)	Configure study-dependent input	Larger than threshold	0.001

5 From the **Frequency selection** list, choose **List**.

6 In the **List** text field, type `range(77.5, 0.1, 78.5)`.

#### *Uncertainty Quantification 4*

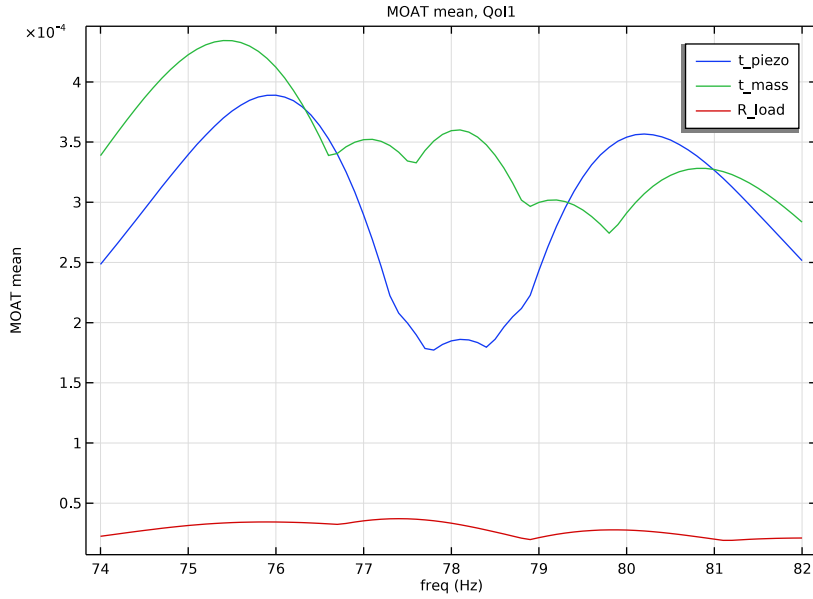
In the **Study** toolbar, click  **Compute**.

To inspect the generated UQ graphs such as MOAT Mean, Sobol Index, and Kernel Density Estimation (KDE) mean, follow these steps:

## RESULTS

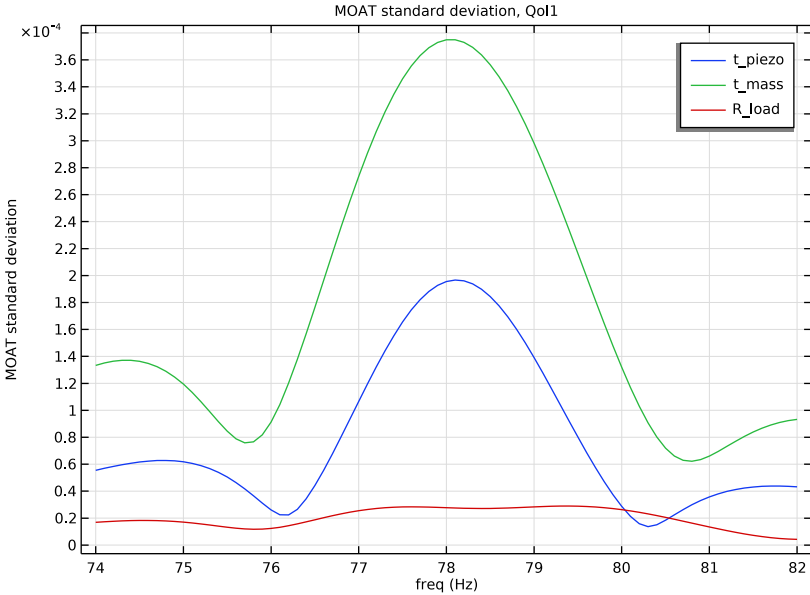
### *MOAT mean, QoI1*

In the **Model Builder** window, under **Results > Uncertainty Quantification Graph** click **MOAT mean, QoI1**.



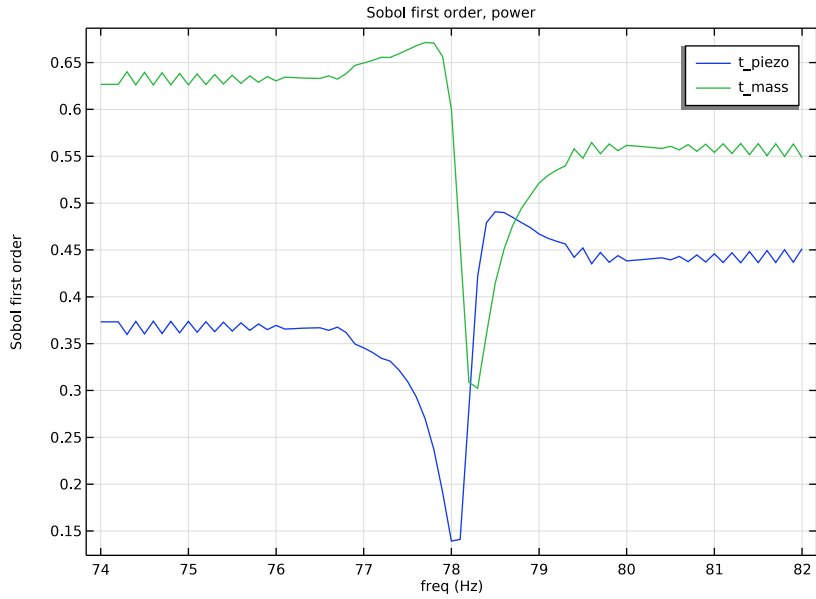
MOAT standard deviation, Qo1

In the **Model Builder** window, click **MOAT standard deviation, Qo1**.



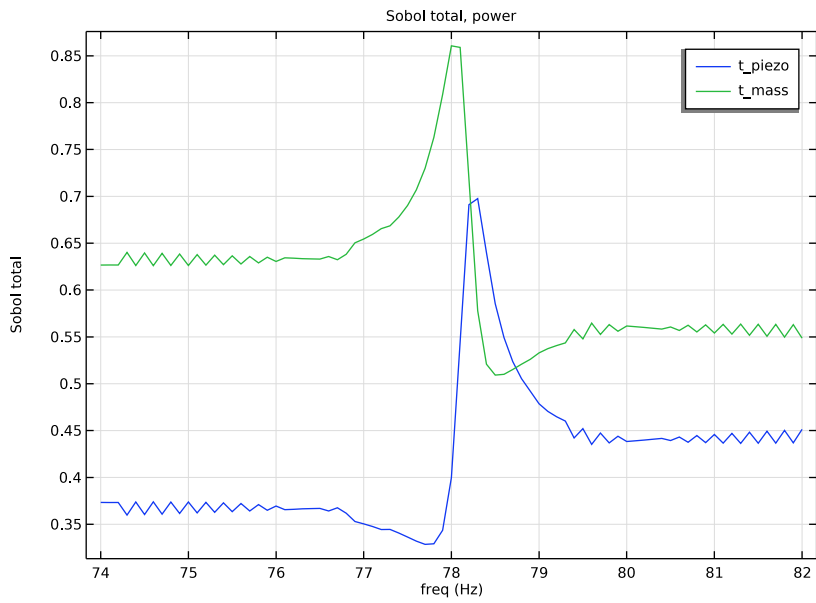
*Sobol first order, power*

In the **Model Builder** window, under **Results > Uncertainty Quantification Graph 1** click **Sobol first order, power**.



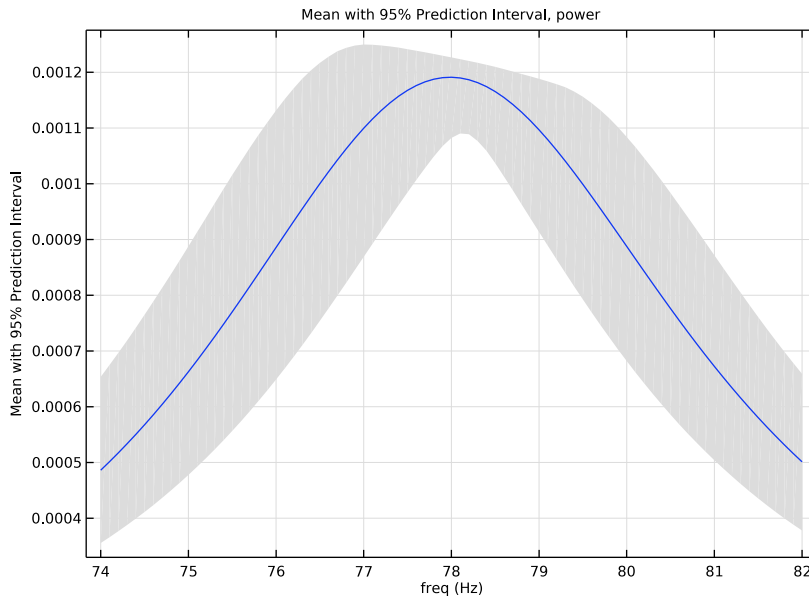
*Sobol total, power*

In the **Model Builder** window, click **Sobol total, power**.



### Mean with 95% Prediction Interval, power

In the **Model Builder** window, under **Results** > **Uncertainty Quantification Graph 2** click **Mean with 95% Prediction Interval, power**.



To create a surface plot of any of the functions from the Polynomial Chaos Expansion surrogate model, take the following steps:

### GLOBAL DEFINITIONS

*Polynomial Chaos Expansion 1 (pce1\_power\_1, pce1\_power\_2, ...)*

- 1 In the **Model Builder** window, under **Global Definitions** click **Polynomial Chaos Expansion 1 (pce1\_power\_1, pce1\_power\_2, ...)**.
- 2 In the **Settings** window for **Polynomial Chaos Expansion**, click to expand the **Plot Parameters** section.
- 3 From the **Function name** list, choose **pce1\_power\_11**.

4 Click  Plot.

