Piezoelectric Energy Harvester

Introduction

The development of extremely low power electronics and wireless systems has led to a strong interest in the field of energy harvesting — the development of miniature generators. Typically these devices are used to power sensors and wireless communication systems, enabling standalone 'wireless sensors' that are cheap to deploy. Frequently wireless sensors make measurements intermittently over an extended period, reporting via a wireless link to other sensors and ultimately to a base station that records readings from all the deployed sensors (creating a 'wireless sensor network'). This model analyzes a simple "seismic" energy harvester, that is designed to generate electrical energy from local variations in acceleration, that occur, for example, when a wireless sensor is mounted on a vibrating piece of machinery. The energy harvester analyzed in this model consists of a piezoelectric bimorph clamped at one end to the vibrating machinery with a proof mass mounted on its other end. It is loosely based on the mechanical system described in detail in Ref. 1.

Model Definition

Figure 1 shows the device geometry. The power harvester consists of a piezoelectric bimorph clamped at one end with a proof mass mounted on the other end. The bimorph has a ground electrode embedded within it (coincident with the neutral plane of the beam) and two electrodes on the exterior surfaces of the cantilever beam. This configuration ensures that same voltage is induced on the exterior electrodes, even though the stress above and below the neutral layer is of opposite sign. Since the clamp is mounted to a piece of vibrating machinery the device is analyzed in a vibrating reference frame (modeled in COMSOL by the application of a sinusoidal body load).

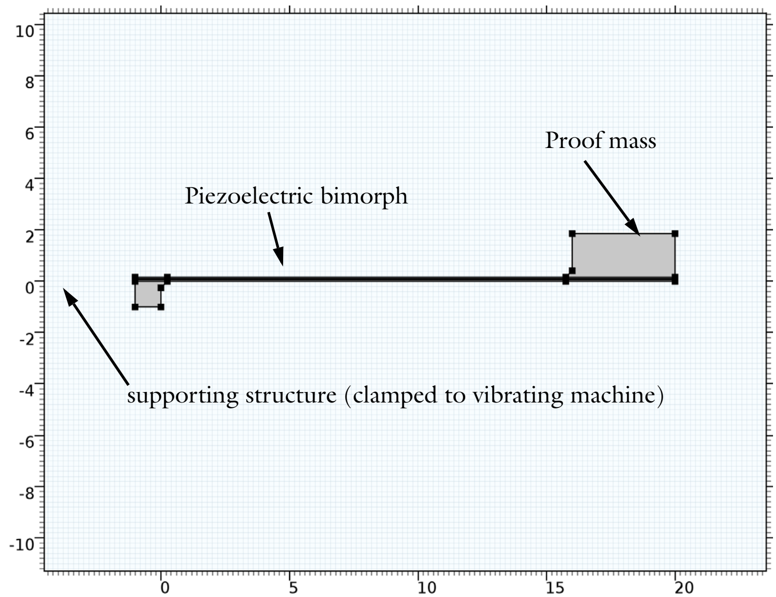


Figure 1: 2D model geometry, showing the major components of the energy harvester, including the piezoelectric bimorph, proof mass and supporting structure.

The model performs three analyses of the mechanical part of the energy harvester system. First, the power output is analyzed as a function of vibration frequency, with a fixed electrical load. Then the power output as a function of electrical load is explored. Finally the DC voltage output, as a function of acceleration, is shown to be linear.

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 kΩ. The response of the system shows a peak at 76Hz, close to the computed resonant frequency of the cantilever at 73 Hz (from a separate eigenfrequency analysis of this device).

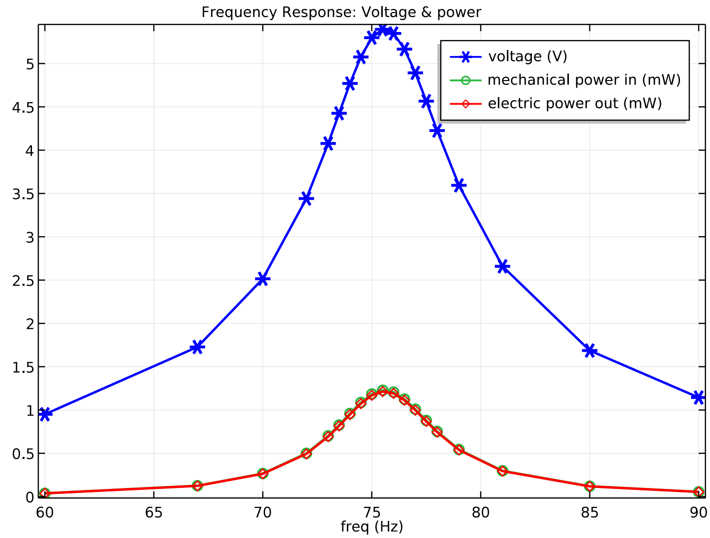


*Figure 2: Energy harvester input mechanical power and the power harvested (in mW) as well as the peak voltage induced across the piezoelectric bimorph (in V) vs. excitation frequency. The load impedance is 12 k*Ω *and the acceleration magnitude is 1 g.*

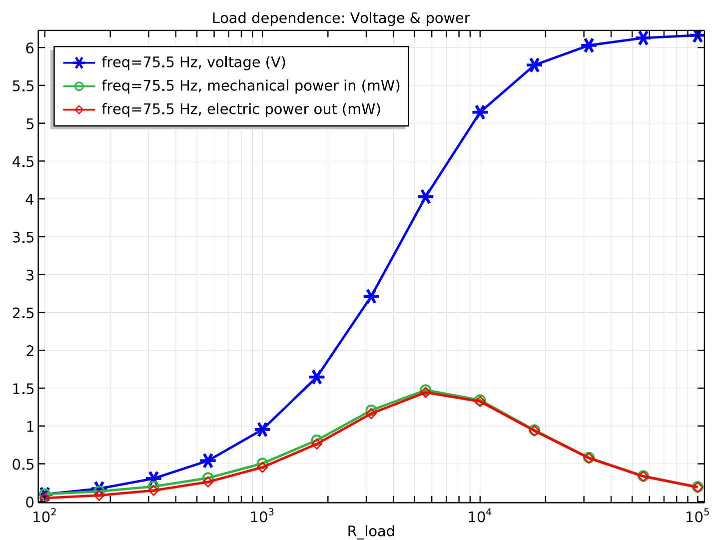


Figure 3: Power harvested from the device as a function of the electrical load resistance at an acceleration of 1 g oscillating at 75.5Hz.

Figure 3 shows the harvested power from the device as a function of the electrical load resistance at an acceleration of 1 g oscillating at 75.5Hz. The peak in energy harvested corresponds to an electrical load of 6 kΩ.

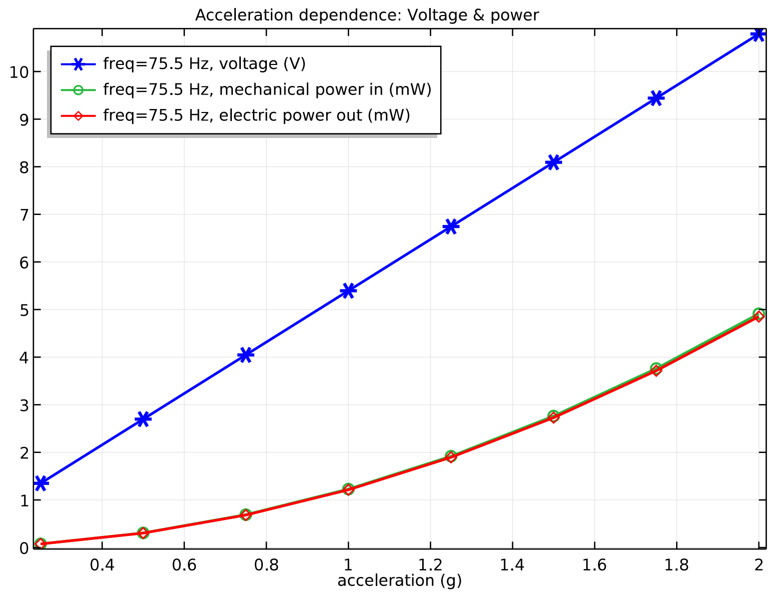


Figure 4: DC voltage and mechanical/electrical power output versus the magnitude of the mechanical acceleration at a fixed frequency of 75.5Hz with a load impedance of 12 kΩ.

Figure 4 shows the DC voltage and mechanical/electrical power output versus the magnitude of the mechanical acceleration at a fixed frequency of 75.5Hz with a load impedance of 12 kΩ. The voltage increases linearly with the load, whilst the harvested power increases quadratically, as expected from equation 4 in Ref. 1.

Note that these results are in good qualitative agreement with those presented in Ref. 1. Completely quantitative agreement would not be expected from a two dimensional model.

References

1. 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, 2007.

Application Library path: `MEMS_Module/Piezoelectric_Devices/piezoelectric_energy_harvester`

Modeling Instructions

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

NEW

1 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>Piezoelectric Devices**.

3 Click **Add**.

4 In the **Select physics** tree, select **AC/DC>Electrical Circuit (cir)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select study** tree, select **Preset Studies for Selected Physics Interfaces>Frequency Domain**.

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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 (g)
R_load	12[kohm]	12000 Ω	Load resistance
w_plate	14[mm]	0.014 m	Out of plane dimension

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**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Position** section.

3 In the **x** text field, type -1.

4 In the **y** text field, type -1.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type 21.

4 In the **Height** text field, type 0.16.

5 Locate the **Position** section. In the **x** text field, type -1.

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

Layer name	Thickness (mm)
Layer 1	0.06
Layer 2	0.04

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type 4.

4 In the **Height** text field, type 1.7.

5 Locate the **Position** section. In the **x** text field, type 16.

6 In the **y** text field, type 0.16.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **r1** and **r2** only.

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

4 Under **Input_objects**, click **Zoom to Selection**.

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

6 Clear the **Keep interior boundaries** check box.

Add chamfers to the model to avoid stress singularities at the reentrant corners.

Chamfer 1 (cha1)

1 On the **Geometry** toolbar, click **Chamfer**.

2 On the object **uni1**, select Points 7 and 8 only.

3 In the **Settings** window for Chamfer, locate the **Distance** section.

4 In the **Distance from vertex** text field, type `0.25`.

Rectangle 2 (r2)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Rectangle 2 (r2)** and choose **Duplicate**.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **x** to `0.25`.

5 In row **2**, set **x** to `0.25`.

6 In row **2**, set **y** to `0.16`.

Bézier Polygon 2 (b2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **x** to `15.75`.

5 In row **2**, set **x** to `15.75`.

6 In row **1**, set **y** to `0.16`.

7 On the **Geometry** toolbar, click **Build All**.

MATERIALS

On the **Home** toolbar, click **Windows** and choose **Add Material**.

ADD MATERIAL

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

2 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-5A)**.

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

ADD MATERIAL

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

2 In the tree, select **Built-In>Structural steel**.

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

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

MATERIALS

Structural steel (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Structural steel (mat2)**.

2 Select Domains 1, 3, 6, 9, and 11 only.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

Damping 1

1 On the **Physics** toolbar, click **Attributes** and choose **Mechanical 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 η_s list, choose **User defined**. In the associated text field, type `0.001`.

Piezoelectric Material 1

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

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

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

4 Select Domains 2, 4, 5, 7, 8, and 10 only.

Mechanical Damping 1

1 On 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 `0.001`.

Body Load 1

1 On 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

0	x
-solid.rho*g_const*acc	y

Fixed Constraint 1

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

2 Select Boundary 2 only.

ELECTROSTATICS (ES)

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

2 Select Domains 2, 4, 5, 7, 8, and 10 only.

3 In the **Settings** window for Electrostatics, locate the **Thickness** section.

4 In the d text field, type `w_plate`.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundaries 4, 9, 13, 18, 20, and 26 only.

Terminal 1

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

2 Select Boundaries 6, 8, 15, 17, 22, and 24 only.

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

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

ELECTRICAL CIRCUIT (CIR)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electrical Circuit (cir)**.

Resistor 1

1 On 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

1 On 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)**.

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**.

Mapped 1

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Mapped**.
2 In the **Settings** window for Mapped, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domains 5–7 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.
2 Select Boundaries 19 and 23 only.
3 In the **Settings** window for Distribution, locate the **Distribution** section.
4 In the **Number of elements** text field, type `3`.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Boundary 21 only.

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

4 In the **Number of elements** text field, type 2.

Distribution 3

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

2 Select Boundaries 13, 15, 17, and 18 only.

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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 30.

6 In the **Element ratio** text field, type 20.

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

Mapped 2

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

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

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

4 Select Domains 8–10 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 2** and choose **Distribution**.

2 Select Boundaries 20, 22, 24, and 26 only.

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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

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

6 In the **Element ratio** text field, type 20.

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

Mapped 3

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

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

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

4 Select Domains 2–4 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 3** and choose **Distribution**.

2 Select Boundaries 4, 6, 8, and 9 only.

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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

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

6 In the **Element ratio** text field, type 10.

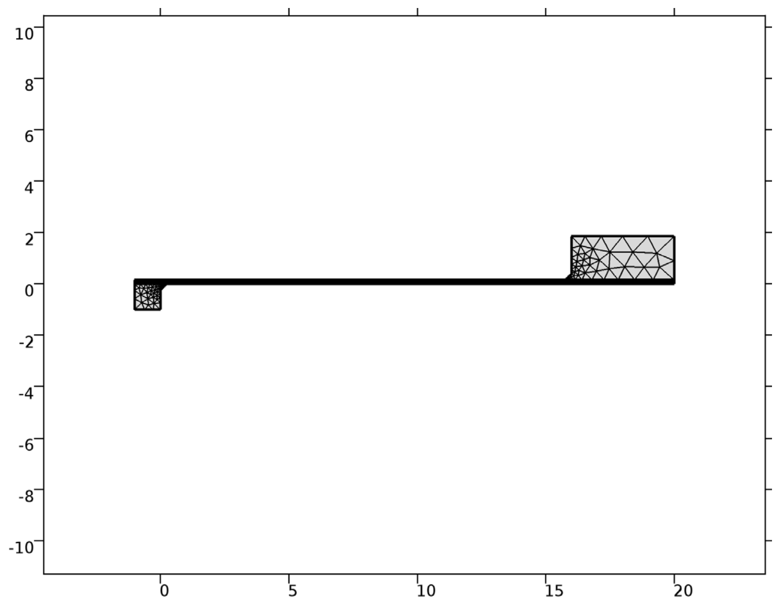
Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Move Down**.

2 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Move Down**.

3 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Move Down**.

4 On 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 `Frequency Response` in the **Label** text field.

FREQUENCY RESPONSE

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Frequency Response** 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 `60 67 70 72 range(73,.5,78) 79 81 85 90`.

Disable direct solver error checking which is too stringent in this case.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

DEFINITIONS

Define an integration operator to calculate mechanical power input later when plotting results.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

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

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

FREQUENCY RESPONSE

Solution 1 (sol1)

1 In the **Model Builder** window, under **Frequency Response>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** click **Direct**.

2 In the **Settings** window for Direct, click to expand the **Error** section.

3 From the **Check error estimate** list, choose **No**.

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

RESULTS

1D Plot Group 3

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

2 In the **Settings** window for 1D Plot Group, type `Frequency Response: Voltage & 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 & power`.

Global 1

On the **Frequency Response: Voltage & power** toolbar, click **Global**.

Frequency Response: Voltage & power

1 In the **Settings** window for Global, locate the **y-Axis Data** section.

2 In the table, enter the following settings:

Expression	Unit	Description
abs(cir.R1_v)	V	voltage (V)
0.5*intop1(realdot(-solid.rho*g_const*acc,solid.u_tY))*w_plate	mW	mechanical power in (mW)
0.5*realdot(cir.R1_i,cir.R1_v)	mW	electric power out (mW)

3 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 From the **Positioning** list, choose **In data points**.

6 On the **Frequency Response: Voltage & power** toolbar, click **Plot**.

ROOT

On the **Home** toolbar, click **Windows** and choose **Add Study**.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select study** tree, select **Preset Studies**.

3 In the **Select study** tree, select **Preset Studies>Frequency Domain**.

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

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

STUDY 2

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2** 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 `75.5`.

4 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
R_load	10^range(2,0.25,5)	

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

8 In the **Settings** window for Study, type `Load dependence` in the **Label** text field.

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

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

LOAD DEPENDENCE

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

2 In the **Model Builder** window, expand the **Load dependence>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node, then click **Direct**.

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

4 From the **Check error estimate** list, choose **No**.

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

RESULTS

Frequency Response: Voltage & power 1

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

2 In the **Settings** window for 1D Plot Group, type `Load dependence: Voltage & power` in the **Label** text field.

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

Load dependence: Voltage & power

1 Locate the **Title** section. In the **Title** text area, type `Load dependence: Voltage & power`.

2 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

3 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

4 On the **Load dependence: Voltage & power** toolbar, click **Plot**.

ROOT

On the **Home** toolbar, click **Windows** and choose **Add Study**.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select study** tree, select **Preset Studies>Frequency Domain**.

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

4 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Results>Load dependence: Voltage & power** node, then click **Study 3>Step 1: Frequency Domain**.

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

3 In the **Frequencies** text field, type `75.5`.

4 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
acc	range(0.25,0.25,2)	

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

8 In the **Settings** window for Study, type `Acceleration dependence` in the **Label** text field.

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

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

ACCELERATION DEPENDENCE

Solution 3 (sol3)

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

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

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

4 From the **Check error estimate** list, choose **No**.

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

RESULTS

Load dependence: Voltage & power 1

1 In the **Model Builder** window, under **Results** right-click **Load dependence: Voltage & power** and choose **Duplicate**.

2 In the **Settings** window for 1D Plot Group, type `Acceleration dependence: Voltage & power` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Acceleration dependence/ Solution 3 (sol3)**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `acceleration (g)`.

6 Locate the **Axis** section. Clear the **x-axis log scale** check box.

7 Locate the **Title** section. In the **Title** text area, type `Acceleration dependence: Voltage & power`.

8 On the **Acceleration dependence: Voltage & power** toolbar, click **Plot**.