

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).

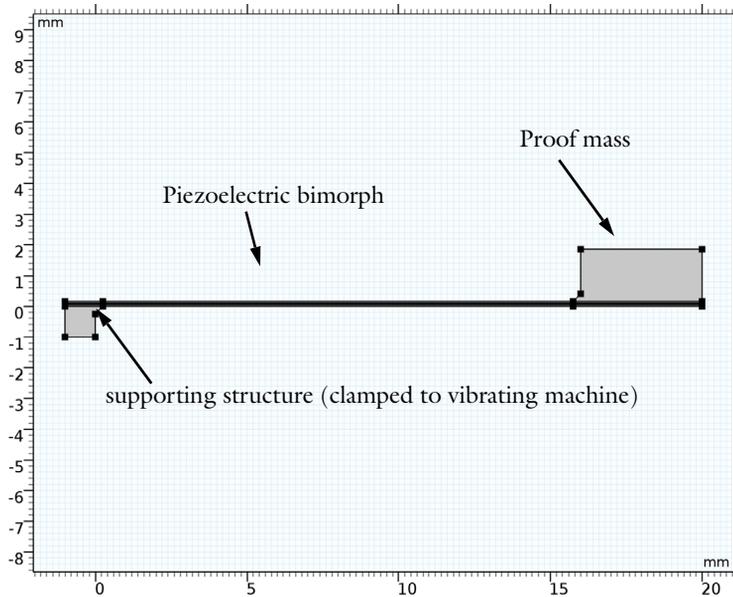


Figure 1: 2D model geometry, showing the major components of the energy harvester, including the piezoelectric bimorph, proof mass and supporting structure.

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 k Ω . 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).

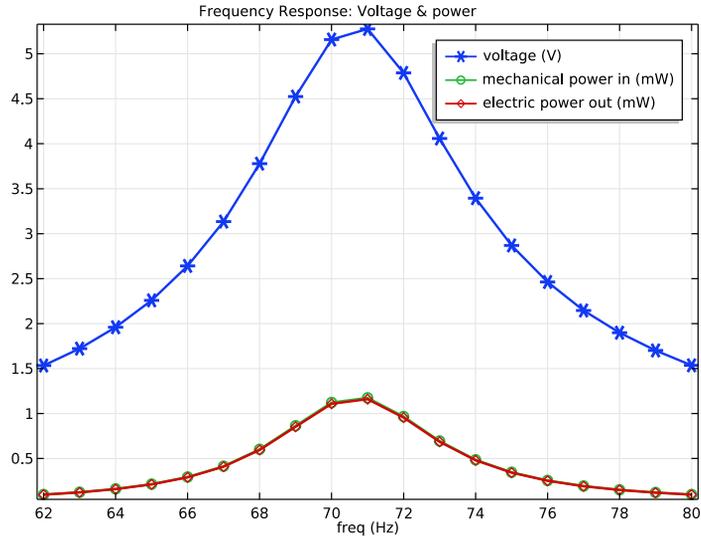


Figure 2: Energy harvester input mechanical power and the power harvested (in mW) as well as the peak voltage induced across the piezoelectric bimorph (in V) vs. excitation frequency. The load impedance is $12\text{ k}\Omega$ and the acceleration magnitude is 1 g .

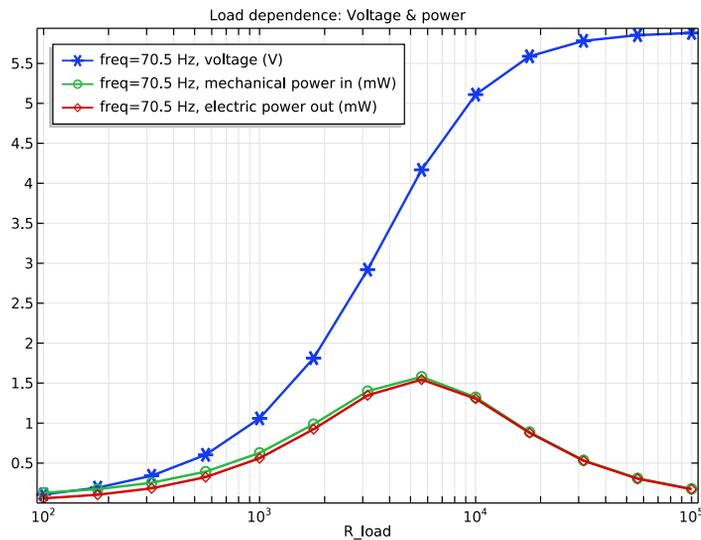


Figure 3: Power harvested from the device as a function of the electrical load resistance at an acceleration of 1 g oscillating at 75.5 Hz .

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 k Ω .

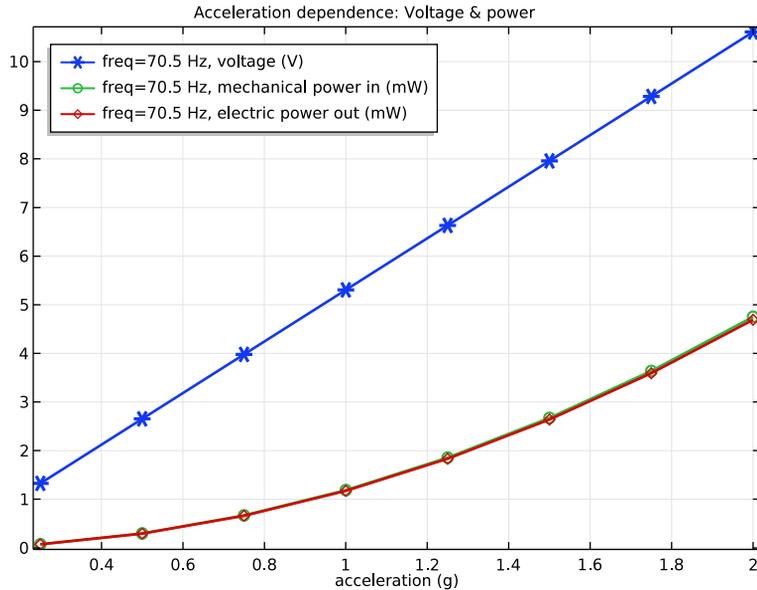


Figure 4: DC voltage and mechanical/electrical power output versus the magnitude of the mechanical acceleration at a fixed frequency of 75.5Hz with a load impedance of 12 k Ω

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 k Ω . 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

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 In the **Model Builder** window, under **Global Definitions** 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 (un1)

- 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 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.

Bézier Polygon 1 (b1)

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

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

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

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

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

6 In row **2**, set **x** to 0.25 and **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 and **y** to 0.16.

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

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

ADD MATERIAL

1 On 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>Lead Zirconate Titanate (PZT-5A)**.

4 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)

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 **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 R1

- 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)**.

The high aspect ratio of the modeled geometry makes this problem numerically challenging. There is only a moderate range of mesh sizes where the result is reliable within a few percent. Outside of this range, with the mesh either too coarse or too fine, the result is not reliable.

MESH 1

Size

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 0.02.
- 5 In the **Minimum element size** text field, type 0.002.

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 range (62, 1, 80).
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 Component Coupling** 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

ID Plot Group 3

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID 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

- 1 Right-click **Frequency Response: Voltage & 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 |
|---|------|--------------------------|
| $\text{abs}(\text{cir.R1_v})$ | V | voltage (V) |
| $0.5 * \text{intop1}(\text{realdot}(-\text{solid.rho} * \text{g_const} * \text{acc}, \text{solid.u_tY})) * \text{w_plate}$ | mW | mechanical power in (mW) |
| $0.5 * \text{realdot}(\text{cir.R1_i}, \text{cir.R1_v})$ | mW | electric power out (mW) |

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

- 5 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 6 From the **Positioning** list, choose **In data points**.
- 7 On the **Frequency Response: Voltage & power** toolbar, click **Plot**.

ADD STUDY

- 1 On 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 **Preset Studies**.
- 4 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.
- 5 Click **Add Study** in the window toolbar.
- 6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Frequency Domain

- 1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
- 2 In the **Frequencies** text field, type 70.5.
- 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 |
|--------------------------|----------------------|----------------|
| R_load (Load resistance) | 10^range(2,0.25,5) | |

- 6 In the **Model Builder** window, click **Study 2**.
- 7 In the **Settings** window for **Study**, type Load dependence in the **Label** text field.
- 8 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Solution 2 (sol2)

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 **ID 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**.

ADD STUDY

- 1 On 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 **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 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 70.5.
- 3 Locate 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 |
|----------------|-----------------------|----------------|
| acc | range (0.25, 0.25, 2) | |

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

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

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

Solution 3 (sol3)

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 **ID 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**.